University of Southern Queensland

Faculty of Engineering & Surveying

Numerical Analysis of a Closed Face Impeller

A dissertation submitted by

A. Tuxford

in fulfilment of the requirements of

ENG4112 Research Project

towards the degree of

Bachelor of Engineering(Mechanical)

Submitted: October, 2006

Abstract

The flow conditions within the fluid passages of a closed face impeller is difficult to visualise without the application of numerical analysis to simulate these flows in a computer based model. Particle Image Velocimetry (PIV) and Laser Doppler Velocimetry (LDV) can give us some insight into the flows at the inlet and the outlet of the impeller, but this is of limited use as it doesn't show what is happening behind the shroud. The impeller that is being analysed has been provided by Tyco Flow Control/Pumping Systems, and is from their 125x100-500 ISO Pump running at 4 Pole speed (1482 RPM). The impeller has been modelled in 2d using the original AutoCAD drawings and Pro/Engineer to produce a file that can be read in Gambit. After meshing in Gambit, the model was solved using the commercial package Fluent to solve the 2D Reynolds Averaged Navier Stokes equations in a rotating polar co-ordinate system. The flow conditions present in the impeller confirmed some of the suspicions about the orignal design, but some unexpected flow conditions were found, which provides insight into some phenomena seen in service.

University of Southern Queensland

Faculty of Engineering and Surveying

ENG4111/2 ***Research Project***

Limitations of Use

Certification of Dissertation

I certify that the ideas, designs and experimental work, results, analyses and conclusions set out in this dissertation are entirely my own effort, except where otherwise indicated and acknowledged.

I further certify that the work is original and has not been previously submitted for assessment in any other course or institution, except where specifically stated.

A. Tuxford

Q98230156

Signature

Date

Acknowledgments

I would like to extend my thanks to my employers Tyco Flow Control Pacific/Pumping Systems for the opportunity to undertake this research into their product. I would also like to thank Mr. Geoff Sheedy and Mr. Ian Smith for their valuable support and knowledge that has been so useful during this project. My supervisor Dr. Ruth Mossad has provided me with technical knowledge which has proven invaluable and I thank her for this assistance. Lastly I'd like to thank my family and friends for their support and help in bringing this dissertation together. All of the support and assistance has been of great value.

A. Tuxford

University of Southern Queensland

October 2006

Contents

List of Figures

List of Tables

Nomenclature

Acronyms

CAD	Computer Aided Design
CFD	Computation Fluid Dynamics
CNC	Computer Numerical Control
QA	Quality Assurance
$R\&D$	Research and Development
TPS	Tyco Flow Control/Pumping Systems

Units

A	Area (m^2)
$\vec{a}$	Acceleration (m/s^2)
a"	Local speed of sound (m/s)
D_H	Hydraulic diameter (m)
$\vec{F}$	Force vector (N)
$\vec{g}$	Gravitational acceleration (m/s^2) standard values = 9.80665 m/s^2
ℓ, l, L	Length scale (m)
m	Mass (kg)
$\dot{m}$	Mass flow rate (kg/s)
p	Pressure (Pa)
r	Radius (m)
Re	Reynolds number $\equiv$ ratio of inertial forces to viscous forces (dimensionless)

T	Temperature (K)
t	Time (s)
U	Free-stream velocity (m/s)
u, v, w	Velocity magnitude (m/s)
V	Volume (m^3)
$\vec{v}$	Overall velocity vector (m/s)
ϵ	Turbulent dissipation rate (m^2/s^3)
μ	Dynamic viscosity (Pa-s)
ν	Kinematic viscosity (m^2/s)
ρ	Density (kg/m^3)
$\overline{\overline{\tau}}$	Stress tensor (Pa)
τ	Shear stress (Pa)
τ	Time scale (s)
Ω	Angular velocity

Chapter 1

Introduction

Before the advent of computer modelling and analysis, the field of pump design mainly relied upon empirical data collected from experimentation that was conducted mostly in the years following the Second World War. The book which is considered the "Bible" of pump design was written in 1948 by Alexey Stepanoff, and has only had minor changes over the years. The basic design information contained in this book is still in use today, nearly 60 years later. Subsequent books explaining pump design have merely re-stated the information from Stepanoff's book. Books like "Design and Performance of Centrifugal and Axial Flow Pumps and Compressors" (Kovats 1964), "Impeller Pumps" (Stephen Lazarkiewicz & Adam T. Troskolanski 1965) and "Centrifugal and Other Rotodynamic Pumps" (Addison 1966) all reference the equations developed by Stepanoff regarding the design of impellers. It wasn't until many years later that books detailing pump design started to expand upon the work done by Stepanoff all those years ago. Books like "Rotodynamic Pump Design" (Turton 1994) and "Hydrodynamics of Pumps" (Brennen 1994) looked, not at the basic tenants of the design of the pumps, but at refining the performance and efficiency of the pump. These new books examined the impact that cavitation had upon the impeller and also what could be achieved with more sophisticated manufacturing methods.

With the rapid increases in computing power in the past decade, the ability of pump designers and researchers to analyse and gather data from previously un-accessible areas of pumps via tools such as computational fluid dynamics (CFD) has lead to development of pumps which are capable of operating in fields that are new and under-developed. The simulation of pumping blood and the visualisation of the damage inflicted by the pump impeller on the blood cells has only been made possible using CFD techniques. While these new techniques are valuable to the pumping industry, there is perhaps one benefit that will see CFD become the design tool of choice for the pumping industry, and that is the ability to perform research and development (R&D) in a virtual environment. By designing, modelling and then testing wholly within the virtual environment, the product development times become smaller and the company utilising these tools can react better to the constantly changing marketplace.

1.1 Overview

The aim of this project is to provide a proof of concept to the feasibility of using computer based tools as a way to cut the costs of research and development for my employers. All previous R&D projects have been achieved using trial and error methods and intuition, which with the competitiveness of the current market place, this is no longer enough to keep them at the cutting edge of pump design. My employers hope that this simplified simulation will confirm the usefulness of using computer based tools in pump development, and will allow us to move onto more complex modelling as required.

1.2 Objectives

The objective of this project was to conduct a numerical analysis on a simplified two dimensional (2D) impeller selected by my project sponsor and employer. The numerical analysis of the chosen impeller provided a valuable insight into the flow conditions that are present inside the shrouded area of the impeller. The major points of interest to the company when looking at this simplified model were:

1. Separation: Separation can be identified where the fluid loses contact with the walls of the fluid passage. This loss of contact produces an area of little to no

flow and can also produce a backflow;

2. Turbulence: This is often a by product of separation occurring. If the separation is early in the passage, this can upset the flow downstream to the extent that instead of smoothly flowing, the fluid tumbles and swirls losing energy, and

3. Velocity: The velocity of the fluid of the impeller should exit at a uniform velocity (Stepanoff 1957, 31). This is an important consideration as the impeller is the only device in the pump that contributes to the head generated. All the other components only contribute to losses in the pump.

These three points of interest can all contribute to the losses in an impeller, but the biggest losses occur due to the separation/turbulence. As the impeller takes the rotary motion of the pump shaft and converts this into radial velocity at its outlet, anything that disrupts this acceleration will have an impact on the head generated by the pump.

1.3 Sponsor Information

Tyco Flow Control is a premier manufacturer and developer of world class products and services to the water industry. From pumps, tanks and travelling irrigation through to valves , ABS pipes systems, measurement and control equipment and product servicing and design we meet your exacting standards.

Through the various divisions we cater for specialist pumping requirements, as well as your day to day supply. Tyco Pumping Systems provides sales, maintenance, repair and design of your pumping systems requirements, providing a required industry service.

Southern Cross is an Australian icon in the pumping and Irrigation industry having been established since 1871. The development of Southern Cross windmills and associated water storage and handling equipment played an important part in Australia's early development. The success of our early pioneering settlers basically came back to a dependence on one vitally important resource.... a reliable supply of good quality water. Today's modern industrial, community and agricultural developments have that same reliance, and even greater demand, for safe, clean and reliable supplies of water.

Water handling equipment has been, and still is, the central development and manufacturing focus for Southern Cross throughout the 135 years, along with the marketing, distribution and servicing of the products not only throughout Australia but also to many developing countries worldwide.

In the early development the company also manufactured an extended range of products including such things as wool presses, steel railway sleepers, steam boilers, stationary and portable steam engines, dairy equipment, railway rolling stock and steam locomotives for both the Queensland and Commonwealth Governments. Quantities of munitions and appliances used during two world wars, and including hundreds of engines, pumps, air compressors, diesel marine engines and machine tools were manufactured for British Forces in North Africa, Americans for use in the East and for our own Army and Navy.

Currently the Southern Cross product range extends from small domestic water supply systems, through broadacre agricultural irrigation schemes, to specialised products for mining, heavy industry and community water supplies. Australia's leading range of ISO Standard centrifugal pumps, close coupled motorpumps, multi stage centrifugal pumps, submersible borehole pumps, computer controlled pump stations, and heavy duty water storage tanks and tankstands are all key products for the company. The irrigation product range includes small self-propelled turf and small crops irrigators, and high performance broadacre travellers.

Everflow Pumps was established in Sydney in 1986 and emerged as a leading manufacturer and supplier of air operated diaphragm pumps, submersible pumps, turbine pumps and axial flow pumps for both the agricultural and industrial markets. Everflow Pumps was acquired in 2000 and later relocated to the new Southern Cross manufacturing facility in Withcott, Queensland. The broad range of Everflow turbine and submersible pumps is now manufactured and distributed from Withcott and marketed under the Everflow Pumps brand.

1.4 Dissertation Overview

Chapter 1 Introduction

Chapter 2 Background Information

Chapter 3 Literature Review

Chapter 4 Mathematical Modelling and Boundary Conditions

Chapter 5 Modelling and Meshing

Chapter 6 Numerical Analysis Procedure, Results and Discussion

Chapter 7 Conclusion

Appendix A Project Specification

Appendix B Pump Curve

Appendix C Fluent Contours

Appendix D Company Brochure

Chapter 1 introduced the topic and explained the objectives and scope of the research. Chapter 2 will provide a brief discussion of the background information, for example the factors that effect current impellers in service, and how these factors are being worked around and what solutions have currently been found. Chapter 3 will summarise the details of the literature review undertaken before carrying out the CFD simulations. Chapter 4 will show the basic equations for the numerical method used by Fluent, and the realisable k-ϵ turbulence models used for this CFD simulation. Chapter 5 discusses some of the limitations and issues encountered during the meshing and modelling phase of the dissertation. Chapter 6 presents the numerical analysis procedure, results and discussion. Finally all of the above will be summarised, along with knowledge acquired during the research into the factors effecting pump design in Chapter 7.

Chapter 2

Background Information

2.1 Chapter Overview

This chapter will briefly cover some of the problems facing pump manufacturers and the solutions that are currently being implemented to address these issues. While some of the issues that have been encountered during the course of my research are confidential and won't be discussed here, many of the problems that are being encountered are industry wide and will be discussed. This chapter will also cover some of the reasons for improving the design of pump impellers so that the company and the customer both benefit.

2.2 Current Issues

Currently there are four main issues which confront the pump manufacturers when producing their products. These issues have been broadly grouped into the following categories:

1. Casting Issues;
2. Machining Issues;

3. Assembly Issues; and

4. Design Issues.

Of these issues, the first three do not require a vast amount of engineering input, and are mainly quality assurance (QA) issues. The final point, design issues, requires significant engineering input to produce a workable solution. It is the design issues this dissertation hopes to illuminate, that will lead to a method of pump development which will enable the engineering resources of the company to be channelled in the most appropriate direction.

Figure 2.1: Centrifugal Pump Impeller. Picture reproduced with permission of Tyco Flow Control/Pumping Systems.

2.2.1 Casting Issues

All of the impellers in use in the pumps produced at Tyco Flow Control/Pumping Systems (TPS) are cast to produce the fluid passages and the basic shape. These raw castings are then machined to the final shape and tolerances, producing the sealing surfaces needed for the pump to operate effectively (Figure 2.1). But due to the fact that the fluid passages are left as cast, the casting quality of the impeller has a huge impact on the performance of the pump as a whole. Should the surface not be smooth and free of defects, the impeller loses it's ability to function as designed and the performance of the pump is reduced. Listed below are some of the common faults encountered:

1. Symmetry - Due to core shift (where the sand core that defines the fluid passage moves), the calculated blade spacing, blade position and shroud thickness can be incorrect. This can cause one blade to stall or produce flow disruptions that effect the blades around it and reduce the performance of the impeller;

2. Surface Finish - The surface of the fluid passages, and the quality of the finish, is one of the problems that is most likely to be encountered in the production of pump impellers. By pouring the metal too hot or too cold, by not using the correct sand or by pouring too fast or too slow, the surface finish can be adversely affected. Also if the mix of sand and binder isn't correct when the cores are made, the surface won't be hard enough to withstand the heat and forces imparted on it by the molten metal and will break up. Small defects like a thin raised line of metal (less than 1mm high $\times$ 1mm wide) across the wall in the impeller fluid passage can effect the performance of the pump by as much as 3%;

3. Finishing - Sometimes the casting may be perfect, but then subsequently let down by the fettling process. The objective of this process is to clean up the raw casting to remove flashing and other surplus material that will be an obstacle to the machining process or won't be touched by the machining process. One of these areas is the blade tips, where due to the casting process, the shape may not be perfect. Problems arise if the fettling is not done carefully enough, and the shape of the blade tips can be left square, rather than rounded for smooth flow. This can have a large impact on the performance of the impeller, but fortunately, this rarely happens due to procedures in place to educate the workers doing the fettling as to the correct shapes.

2.2.2 Machining Issues

The machining of the impellers at TPS is done either via an internal production order, or via an outplant machinist. Typically we have very few issues from impellers machined in-house, but we do encounter some QA issues with the jobs produced through a third party manufacturer. The lack of problems internally can be attributed to the machinists skill levels and familiarity with the product design. Some of the issues encountered externally are:

1. Symmetry & Balance - One of the most common issues encountered with outplant machining is the symmetry and balance of the finished item. As the impellers are machined using computer numerical control (CNC) machinery, there is a tendency to just place casting in the machine and start the machining operation. This can lead to the impeller being machined off centre and this results in both balancing problems and performance issues as the blade tips don't all follow the same path and some are effected by the preceding blades.

2. Tolerances - Probably the most important stage of the machining process of the impeller is the machining of the wear rings on the front and on some impellers, the rear. These wear rings need to have very small clearances to stop the high pressure fluid from the volute making it's way back into the suction side of the pump. Should the machinist incorrectly measure or gauge these surfaces, the efficiency of the whole pump is reduced.

3. Finishing - As with casting, a perfectly machined impeller can be badly compromised by poor finishing. Should the program be incorrect and not producing the correct surface finish on the critical areas, the efficiency of the impeller will be impaired. It is also important that the machinist correctly fettles the machined item to remove any swarf or remaining metal that can block the fluid pathways.

2.2.3 Assembly Issues

When the impellers are taken out of stock to be incorporated into pumps, a number of issues can occur that will impact on the performance of the pump. Due to the skill level and product knowledge of the pump assembly group at TPS, these issues rarely are encountered, but for completeness these issues have been included.

1. Handling - Utmost care needs to be taken when handling the impellers, all the way from the racking until they are finally inside the pump unit. At any stage of the assembly process, they can receive damage that will affect the performance of the pump. The most common form of damage is from dropping or hitting the impeller against something and consequently damaging the machined surfaces, or deforming the impeller itself. The most severe damage is the deformation of the edge or the wear ring area of the impeller, where it can run against the volute and absorb power, thus compromising the efficiency.

2. Cleanliness - Due to the small tolerances employed in the wear ring area of the pump, it is important that all the components of the pump are as clean as possible to ensure that no debris is present which could cause binding or drag between the impeller and the volute. This debris can be something as simple as a small piece of swarf that hasn't been cleaned out of the impeller or volute after machining, and can produce enough drag to overload the motor driving the pump causing a potentially expensive warranty claim.

2.2.4 Design Issues

The field of pump design used to take a very much hands on approach to development. While impeller blade shapes and configurations could be calculated, the impeller's interaction with other parts of the pump, such as the volute and the cut-water or tongue couldn't be evaluated using numerical methods of the day. The verification and optimisation of the design occurred on the test facility where the design engineer would experiment with different configurations to find the optimal combination. As can be seen in Figure 2.2, removing the pump from the pipe work and drive is a time consuming task, and this would have to happen for every experiment that occurred.

While the original design of the impellers used in the pumps produced by TPS are very good, there are some areas that could be improved;

Figure 2.2: Pump Test Facility at Tyco Flow Control/Pumping Systems Southern Cross Manufacturing facility. Picture reproduced with permission of Tyco Flow Control/Pumping Systems.

1. Efficiency - Some of the current impeller designs don't provide efficiency levels as high as the other designs. This issue effects the sales of these units , as with a lower efficiency the power required is higher and this higher power draw makes the pump more expensive to run. This also can sometimes cause the pump to require a larger motor or engine which increases the costs significantly.

2. NPSHR - Another area where improvements can be made is the Net Positive Suction Head (NPSH) required by the pump. Some designs require more pressure on the suction side to prevent cavitation, and this can be limiting in some installations where a suction lift is required.

2.3 Current Solutions

At this point in time, there are solutions to all of the problems that are encountered in the manufacturing process at TPS, except for the design problems, which are to be addressed after the completion of this dissertation. This work is the first step in a larger project to be undertaken at TPS to investigate the current range of centrifugal and mixed flow pumps and optimise the designs.

2.3.1 Casting Issue Solutions

1. Symmetry - To fix the symmetry problems, the patterns and core boxes need to modified to provide positive location via dowels or a similar fixing method to ensure the core can't move during handling or casting.

2. Surface Finish - The surface finish of the impellers can be improved in a number of ways. By addressing the process used to cast the impellers, a better surface finish can be achieved by either investment casting or using better sand casting techniques. A cheaper solution is to apply a coating to the internal surfaces of the impeller, smoothing the rough walls to allow better fluid flow. The drawback of this method is the coating doesn't remain attached to bronze impellers, so it cannot be used on that material type.

3. Finishing - The finishing of the impeller can be improved with better training and education as to what is acceptable and what can cause issues further along the assembly/testing procedure.

2.3.2 Machining Issue Solutions

1. Symmetry & Balance - This issue can be addressed by better quality castings that have better concentricity straight out of the sand, or by pre-machining the castings and ensuring that the surface that is held for the first operation is concentric and correct.

2. Tolerances - The tolerances can be maintained by accurate measurement by a skilled machinist, or by the utilisation of gauges to check that the tolerance is being held within the required values.

3. Finishing - This aspect of the manufacture of the impellers is controlled by a QA process both at the machining and at the pump assembly stages of production.

2.3.3 Assembly Issue Solutions

When the impellers are taken out of stock to be incorporated into pumps, a number of issues can occur that will impact on the performance of the pump. Due to the skill level and product knowledge of the pump assembly group at TPS, these issues rarely are encountered, but for completeness these issues have been included.

1. Handling - These issues can nearly be totally eliminated by providing a workplace that has sufficient lifting tools and conforms to workplace health and safety requirements. Nearly all of the damage inflicted on impellers is a result of trying to maneuver the impeller without the appropriate tools.
2. Cleanliness - This is controlled by ensuring the that the critical areas of the pump are blown clean with compressed air and then double checked by hand before assembly continues.

2.3.4 Design Issue Solutions

The current design issues will be addressed by the continued development of the techniques explored in this dissertation, and by the development of in-house software to allow pump designs to be checked and verified. After the theoretical pump data for the existing range is produced via numerical methods, it will be compared to the existing experimental data from the test facility to provide verification to the methods and modelling techniques used. Once the numerical analysis techniques agree with the experimental data, new development will occur.

2.4 Chapter Summary

The process of finding, analysing and providing corrective actions for production issues is a long and difficult task. By identifying some of the major issues encountered in the production of an impeller, the design process of newer impellers can attempt to solve some of these issues. While some of the problems can't be solved, except by education, sometimes small changes to the design can yield substantially better production items.

Chapter 3

Literature Review

3.1 Chapter Overview

The problem defined by the project specification introduced some difficulty to the literature review process. The literature on the subject focused mainly on the verification of 3D analysis or simulating other flow effects in 3D. While this didn't directly relate to the problem at hand, and didn't allow for any verification, the information contained in the literature was enough to confirm that teh solution was progressing down the correct path.

3.2 Literature

Miner (2000) reported in the Journal of Fluids Engineering that, results from the CFD analysis show good agreement with the experimental data and that the use of coarse grids can still result in accurate information regarding the performance of the pump. This is an important point to make as the ability to use a more coarse mesh while still obtaining results that are valid means that analysis can be performed on normal workstations rather than having to use expensive and difficult to obtain time on large supercomputers to solve a fine mesh.

Muggli (2002) submitted a paper to the American Society of Mechanical Engineers that showed that pump characteristics could be simulated from part load to full load using modern numerical tools, and that these results were in agreement with measurements from the pump, but they stated that "This computational effort is close to the upper limit possible in an industrial environment. With the introduction of new processors for workstations a speed up of a factor of five can be expected and unsteady simulations could become routine."(Pg 3) The solution that was run to simulate the complete pump was done in three dimensions with a fine mesh structure that strained the computational hardware which the researchers had at their disposal. Comparing this result with the information above, it can be seen that CFD is very sensitive to the quality of mesh that is input into the solver. The goal of this project is to produce a mesh that is fine enough to provide the answers which will show potential design issues, but not so fine that it doesn't bring any extra precision to the results and results in more computation time.

Weidong et al. (2003) stated in a paper written for the International Journal of Rotating Machinery that, while the results for the curved blades were in agreement with experimentally produced data, the results for the straight blade impeller didn't correlate well with the experimental data. While the authors of the paper don't explain why the results were not as expected for the straight blades, they confirm that for the curved blades, the results were good. They also state, "The calculation also predicts reasonable results in both the flow pattern and the pressure distribution." (Pg 1) which suggests that CFD will predict the flow patterns and pressures within the impeller with good agreement to experimental data. This paper also mentioned that backflow occurred on the pressure surface when the flow decreased below a certain level, but as this was off the design point of the impeller, it is not going to be considered as anything more than an interesting point to consider when using CFD to test the full performance range of a pump.

Blanco-Marigorta et al.(2000) published a paper for the ASME 2000 Fluids Engineering Divisions Summer Meeting which covers of some of the difficulties in using CFD for pump analysis. The paper speaks of the problems with CFD, turbulence, separation and boundary layer difficulties, along with the complexity of the mesh needed to show the geometry. Most of the problems related to the solution come down to sufficient detail in the mesh at critical areas without causing long computation times.

Hamkins and Bross (2002) investigated using oil films to determine the surface flows present in a pump impeller. They successfully demonstrated that this was a valid technique for flow visualisation, and they stated "They can also be used to adjust boundary conditions for computational fluid dynamics simulations by trial and error until a good match with the measured pattern is found. The resulting computation should be a better representation of the flow than that with a simple boundary condition."(Pg 5). This is an interesting point which will need to be investigated further when the simulation is developed into full three dimensional simulation of pumps. If the boundary conditions of the pump are not met correctly, then the value of the subsequent data is suspect to say the least.

Byskov et al. (2003) have written in their paper for the Journal of Fluids Engineering that the turbulent intensity increases as the flow in an impeller decreases. This increase in turbulence will affect the efficiency and energy applied to the fluid being pumped. This is supported by Coutier-Delgosha et al. (2003) where they found that when the flow dropped to below 50% of the design flow, the numerical solutions accuracy fell markedly. Pedersen et al. (2003) also found by both PIV and LDV methods that at 25% of the design flow, there was a previously unreported phenomenon where alternate fluid passages were stalled and un-stalled. The stalled passages had a large recirculation cell at the inlet blocking the flow. This may go some way to explaining why the CFD results don't correlate with the experimental results, as Byskov et al. didn't find these phenomena in their CFD analysis of the same impeller.

3.3 Chapter Summary

These papers support the fact that using CFD for design analysis is a valid method, providing that validation occurs to prove the modelling and method works.

Chapter 4

Mathematical Modelling and Boundary Conditions

4.1 Chapter Overview

This chapter will cover some of the equations used in pump design and also some of the equations used by CFD applications to provide solutions to the inputs they are given. Due to the vast array of options in configuring CFD applications for a particular solution, the equations covered here will apply only to what was used in this dissertation.

4.2 Impeller and Pump Information

To enable the selection of the correct parameters for the computation of the flow in the fluid passages of the impeller, some of the basic information regarding the pump and where it is operating needs to be collated. To complete this table, some information first needs to calculated.

Firstly we need the specific speed of the pump that is being analysed. Specific speed is a number characterising the type of impeller in a unique manner. Specific speed is determined independent of pump size and can be useful comparing different pump designs. The specific speed identifies the geometrically similarity of pumps.

Specific speed is dimensionless and is expressed as;

$$N_S = \frac{\omega Q^{\frac{1}{2}}}{H^{\frac{3}{4}}} \tag{4.1}$$

Secondly, we need to calculate the Reynolds Number for this pump, to check if the flow through the impeller is expected to be laminar or turbulent. The Reynolds Number is the ratio of inertial forces to viscous forces, and is calculated using the following equation;

$$Re = \frac{\rho v_s D}{\mu} \tag{4.2}$$

Table 4.1: Impeller geometry and pump operating conditions

Geometry			
Inlet Diameter	D_1	145	[mm]
Outlet Diameter	D_2	547	[mm]
Blades	Z	6	[-]
Specific Speed	N_s	604	[-]
Flow Conditions			
$\frac{Q}{Q_d}$		1	[-]
Flow Rate	Q	171	$[\frac{m^3}{h}]$
Head	H	102	[m]
Rotational Speed	η	1480	[rpm]
Reynolds Number	Re	5.51×10^5	[-]

4.3 Governing Equations

4.3.1 Transport Equations

The Reynolds-averaged Navier-Stokes (RANS) equations are used for the basis for all of the analysis work done in this dissertation. These equations govern the transport of the averaged flow quantities, with a wide range of turbulence scales being modelled. The RANS-based modelling approach therefore greatly reduces the required computational effort and resources, and is widely adopted for practical engineering applications (Fluent Inc. n.d., 11.2.1).

The equations for incompressible flow are as follows;

Continuity Equation for rotating flows

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}_r) = 0 \tag{4.3}$$

Momentum Equation for rotating flows with relative velocities

$$\frac{\partial}{\partial t}(\rho \vec{v}_r) + \nabla \cdot (\rho \vec{v}_r \vec{v}_r) + \rho(2\vec{\Omega} \times \vec{v}_r + \vec{\Omega} \times \vec{\Omega} \times \vec{r}) + \rho \frac{\partial \vec{\Omega}}{\partial t} \times \vec{r} \tag{4.4}$$

4.3.2 Turbulence Equations

Because the Reynolds number calculated using equation 4.2 was greater than 1×10^5, the flow will be entering the impeller in a fully developed turbulent state, so a turbulence modelling equation is needed to solve this. By consulting the Fluent technical documentation (Fluent Inc. n.d.), it was determined that a Realisable k-epsilon model (RKE) would be the most appropriate for this type of problem.

> This model has been extensively validated for a wide range of flows, including rotating homogeneous shear flows, free flows including jets and mixing layers, channel and boundary layer flows, and separated flows. For all these

cases, the performance of the model has been found to be substantially better than that of the standard $k - \epsilon$ model. (Fluent Inc. n.d., 11.4.3)

The modelled transport equations for k and ϵ respectively are as follows;

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\epsilon - Y_M + S_k \tag{4.5}$$

and

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_j}(\rho \epsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + \rho C_1 S\epsilon - \rho C_2 \frac{\epsilon^2}{k + \sqrt{\nu\epsilon}} + C_{1\epsilon}\frac{\epsilon}{k}C_{3\epsilon}G_b + S_\epsilon \tag{4.6}$$

4.4 Boundary Conditions

To achieve a valid solution of the model in Fluent, it is very important that the boundary conditions be determined correctly. The most important boundary condition that needs to be met is the impeller inlet and outlet conditions. The pressure drop across the impeller can be found by finding the velocity of the fluid at the inlet, and then using this velocity at the outlet, as Stepanoff states in his book.

So to find the pressure drop, the Bernoulli equation is used. The equation used is given below;

$$p_0 = p_s + \frac{1}{2}\rho v^2 \tag{4.7}$$

Once the pressure drop has been obtained, it is applied to the model in Fluent during the boundary condition configuration stage. This is purely a change in velocity due to the change in fluid passage area, and doesn't account for extra velocity obtained from the impeller's radial motion.

4.5 Chapter Summary

The governing equations for turbulent incompressible rotating flows shown above are the basis for Fluent's solver. These equations are only a small part of the equation set that Fluent calls upon for every solution. To solve these manually would be a huge undertaking for even the simplest of cases, but due the increasing memory and processing power of computers, CFD is becoming a serious tool for the simulation of cases where it is too dangerous to test, the area of interest is hidden or inaccessible to measurement tools or the scale of the simulation prohibits traditional testing due to cost or scaling issues.

Chapter 5

Modelling and Meshing

5.1 Chapter Overview

This chapter aims to describe the processes involved in producing final mesh in Gambit which was then able to be solved in Fluent to arrive at the final numerical solution. This chapter also discusses the refinement of the mesh to converge upon the desired accuracy and the refinement of the boundary layer of the blades to ensure that there is sufficient resolution present to allow accurate modelling. Without good quality meshing, it is often impossible to reach a solution, with the residuals either remaining high or increasing. Once the model has a quality mesh, the solution can be refined more quickly and the ultimate solution reached with a minimum of iterations.

5.2 Modelling in AutoCAD

The AutoCAD model of the impeller was produced from original drawings supplied by TPS for this project. These confidential drawings were re-drawn and simplified to suit the objectives of this dissertation. This simplified model is shown as Figure 5.1 and shows the basic shape of the impeller blades and was drawn at full scale. It is important that the scale and the units are set at this stage of the modelling process, as a mistake here can result in the model scale being incorrect for the subsquent steps of the process.

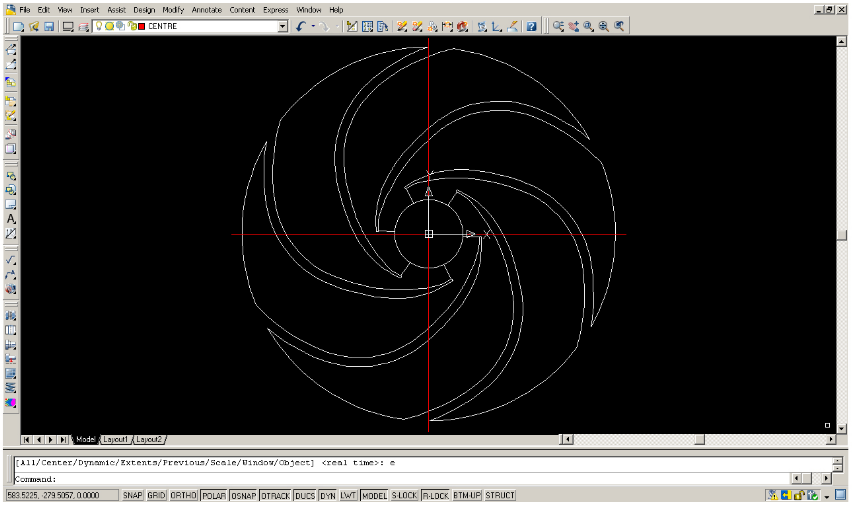


Figure 5.1: Screen capture showing the CAD model completed in AutoCAD

After the basic shape was defined, it was then inspected to ensure that there were no breaks in the lines and that the centre of the impeller was at the origin. At this stage, some additional geometry was added to assist with the meshing process. Once these checks were completed, the model was saved and ready for importing into Pro/Engineer for the next stage of the process.

5.3 Processing in Pro/Engineer

After importing the model data from AutoCAD into Pro/Engineer, the model dimensional parameters were checked to ensure that nothing had changed during the importation. Once the dimensional parameters were verified, the model data was closely inspected to ensure that the blades were fully defined. The importance of checking the model for loops and other geometry abnormalities cannot be stated enough, as the better the quality of the model, the better it can be meshed in Gambit and therefore solved more easily when it is imported into Fluent.

5.4 Importing into GAMBIT

Once the model was finished in Pro/Engineer, it was saved in a format compatible with Gambit and then imported into Gambit for the final stage of geometry creation and the beginning of meshing. As shown in Figure 5.2, the geometry as imported into Gambit has many vertexes (indicated by the + symbols) on the pressure side of the blade, but no mid-span vertexes on any of the other surfaces. This extra data is not needed for such a simple curve, so by implementing the geometry repair functions of Gambit, we arrive at the geometry shown in Figure 5.3. This figure also shows the modifications needed to enable the geometry to be solved. The removal of the larger inlet circle, replacing it with a smaller inlet to more closely replicate the actual inlet conditions is shown. The small straight lines that join the blades to the inlet circle, enable the geometry to be broken up into sections to allow meshing. These blade segments can be seen in Figure 5.4. The outlet of the blades has also been completed here to allow a boundary condition to be applied. This geometry now has all of the necessary items to allow it to be meshed.

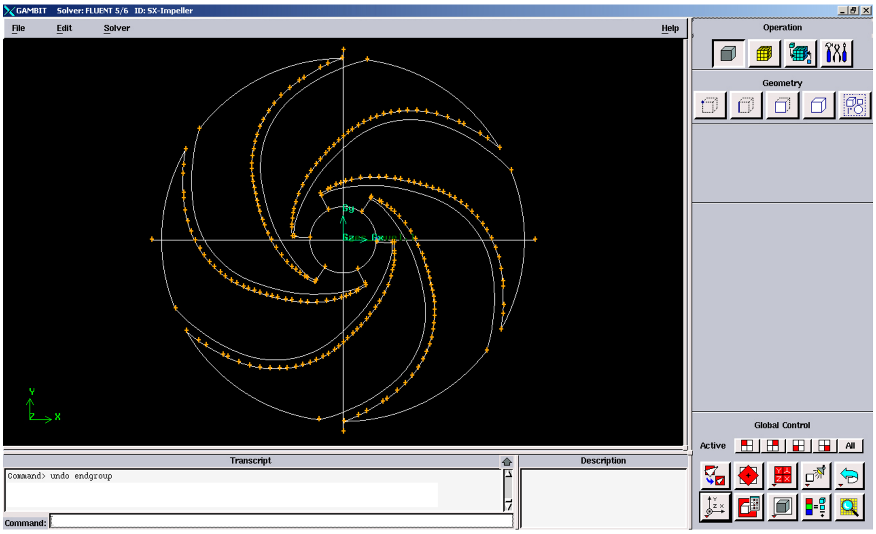


Figure 5.2: Screen capture showing the CAD model as first imported into Gambit

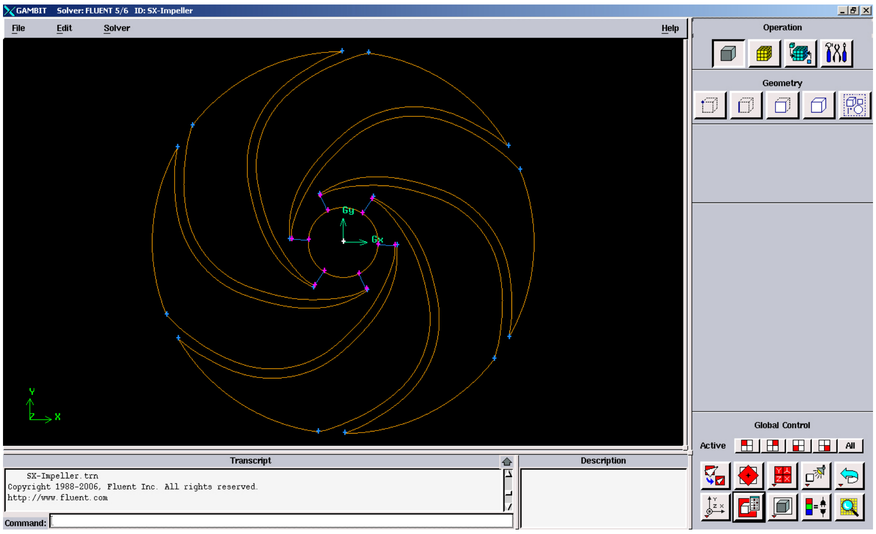


Figure 5.3: Screen capture showing the CAD model after healing and repair in Gambit

5.5 Meshing the Geometry

To mesh the geometry, a mapped quadrilateral mesh was tried first due to the ability of the quadrilateral mesh elements to have a larger aspect ratio than the triangular mesh cells.

From the Fluent 6.2 Documentation on mesh types;

> A characteristic of quadrilateral/hexahedral elements that might make them more economical in some situations is that they permit a much larger aspect ratio than triangular/tetrahedral cells. A large aspect ratio in a triangular/tetrahedral cell will invariably affect the skewness of the cell, which is undesirable as it may impede accuracy and convergence. (Fluent Inc. n.d., 6.1.3)

As can be seen in Figure 5.4 & Figure 5.5, selecting the entire blade region resulted in Gambit meshing over the tip of the blade, and in the process creating a large number of negative regions. This poor first attempt at meshing was discarded and another approach tried.

The next mesh that was tried consisted of the same element type, with a change to the

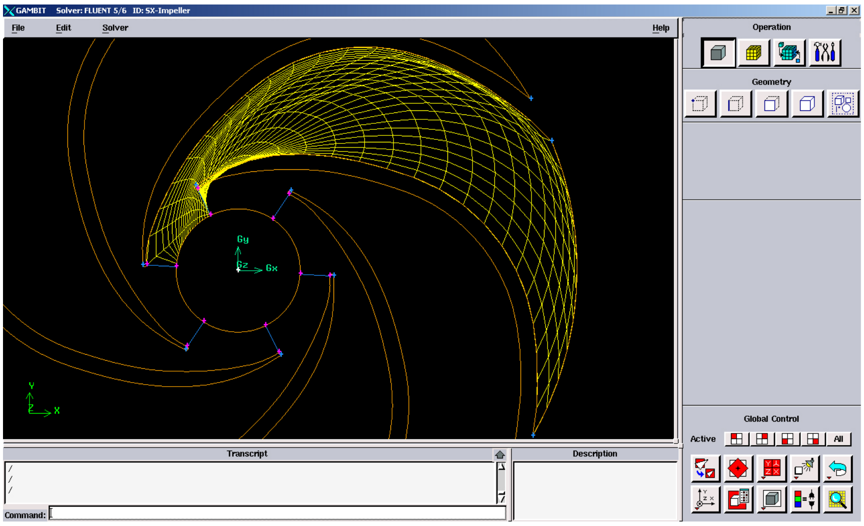


Figure 5.4: Screen capture showing the failed first attempt at meshing

cell size and a fixed number of nodes applied to the very tip of the blade where Gambit had trouble defining the mesh. These nodes can be seen in Figure 5.6. Unfortunately, this approach didn't improve the situation at all, and the resulting mesh as shown in Figure 5.7 clearly shows the same problems as the previous attempt. After attempting to get a mesh that followed the blade profile by using a finer and finer edge node size and failing, I decided to change to a triangular mesh to allow Gambit to resolve blade tip correctly. Using experiences from the previous meshing experiments, it was decided that a coarse triangular mesh would be a sensible starting point to allow the quick generation of the mesh and also allow the initial experimental solutions in Fluent to solve quickly. The nodes used in this mesh are shown in Figure 5.8, and it can be seen that all of the edges in the geometry have had the edge nodes applied so that the face meshing procedure can proceed. Once the geometry was at this stage, it was then just a matter of applying the mapped triangular mesh which can be seen in Figure 5.9 and Figure 5.10.

5.6 Chapter Summary

Despite some initial issues with the meshing in Gambit, the mesh that was produced represented the impeller well and provided sufficient resolution in the areas that were

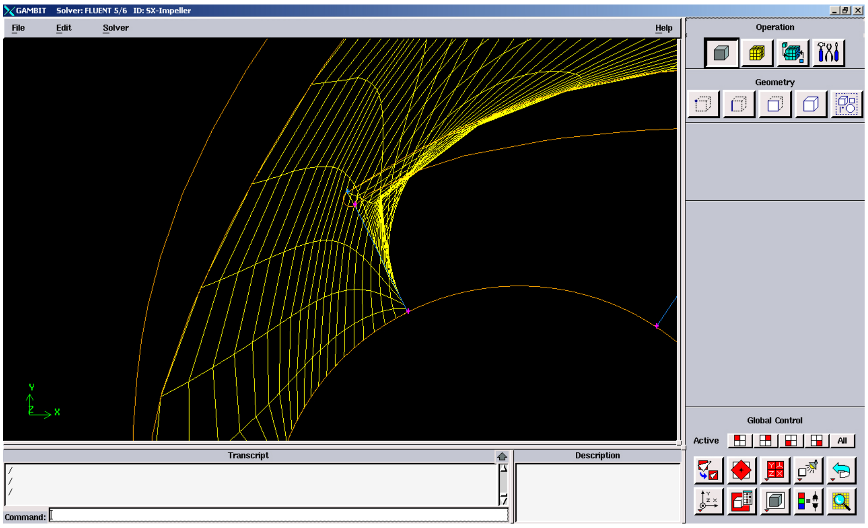


Figure 5.5: Closeup of blade tip mesh

identified as possible problem areas to achieve results that can be used to evaluate the usefulness of CFD as a tool for pump development.

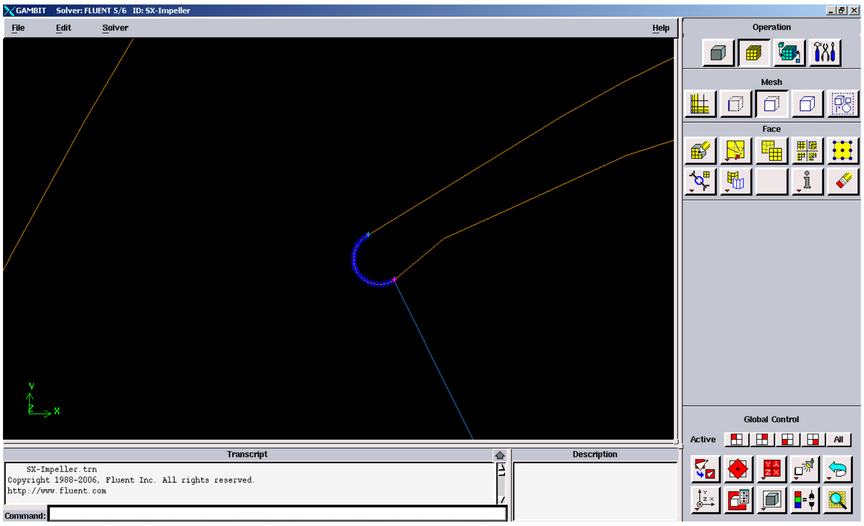


Figure 5.6: Blade tip with fixed nodes

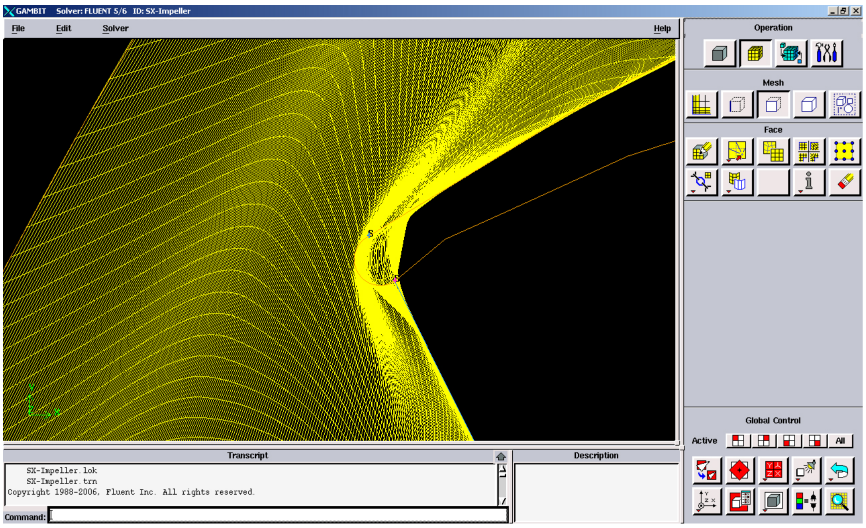


Figure 5.7: Closeup of failed mesh with nodes applied manually to blade tip

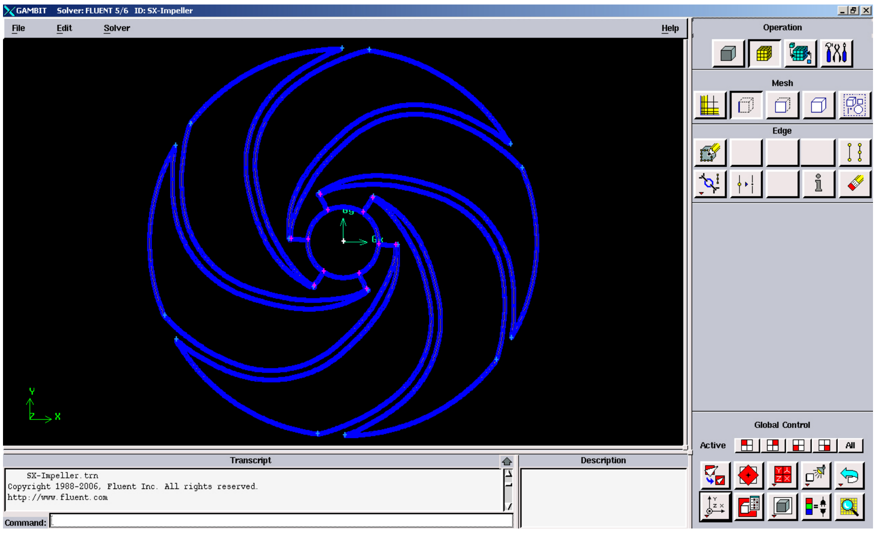


Figure 5.8: Edge nodes on the impeller geometry

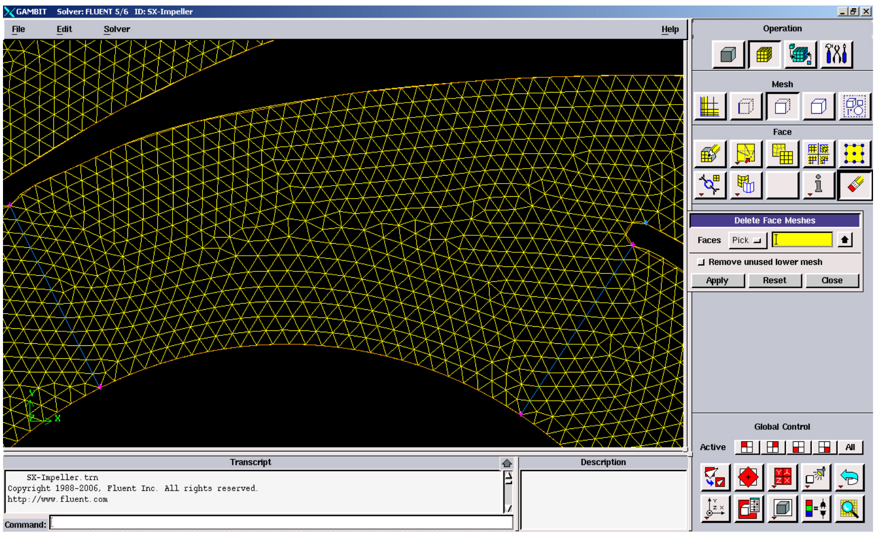


Figure 5.9: Closeup of finished Gambit mesh

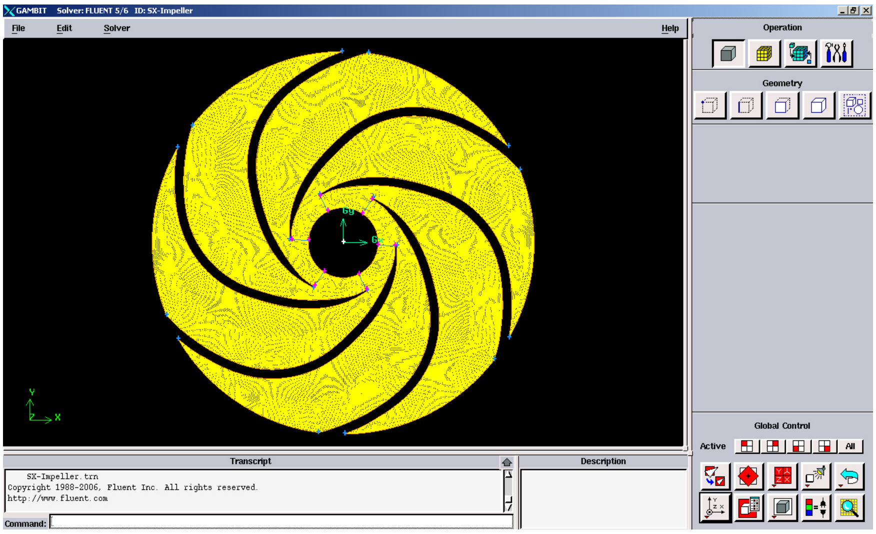


Figure 5.10: Finished Gambit mesh of the whole impeller

Chapter 6

Numerical Analysis Procedure and Results

6.1 Chapter Overview

The following chapter aims to provide an overview of the process necessary to configure Fluent for the solution of the type of problem as defined by this dissertation. The data produced by Fluent will also be presented, with an analysis of these results, discussing the problems encountered and possible solutions.

6.2 Fluent Configuration

When the geometry is first imported into Fluent a number of steps need to be followed to ensure the solver is properly prepared. These steps ensure that all of the solution variables and conditions are configured in a appropriate manner for the solution about to be undertaken, and that the basic geometry and the mesh are suitable.

The following points need to be addressed before a solution run can take place;

1. Unit Systems,

2. Grid Quality,
3. Materials,
4. Solution Type
5. Boundary Conditions,
6. Solution Monitors + Optional Features, and
7. Iterations

These points need to be addressed to ensure the solution can be solved with the greatest accuracy and the least computation time.

6.2.1 Unit Systems

When the geometry is first loaded into Fluent, it has the units that it was created in. If these units are not compatible with the other items loaded, or a change is needed, the scale of the geometry can be adjusted. The geometry created for this dissertation didn't require any scaling, due to consistent length units through its creation.

Fluent was allowed to run with the standard SI unit set for the solution, as difficulties were encountered with the rotating reference frame when the angular velocity was changed from $\frac{rad}{s}$ to RPM.

6.2.2 Grid Quality

The next stage of pre-processing is the inspection of the mesh and the verification of its quality. A visual inspection is carried out by displaying the mesh, and making sure that Fluent had correctly interpreted the Gambit mesh, as is shown in Figure 6.1.

Once this was done, the Smooth/Swap command was executed to ensure that the mesh didn't contain elements that were excessively skewed. Skewed cells in a mesh are a result of incorrect meshing and can cause the solution to diverge and either not solve at all, or crash the solver.

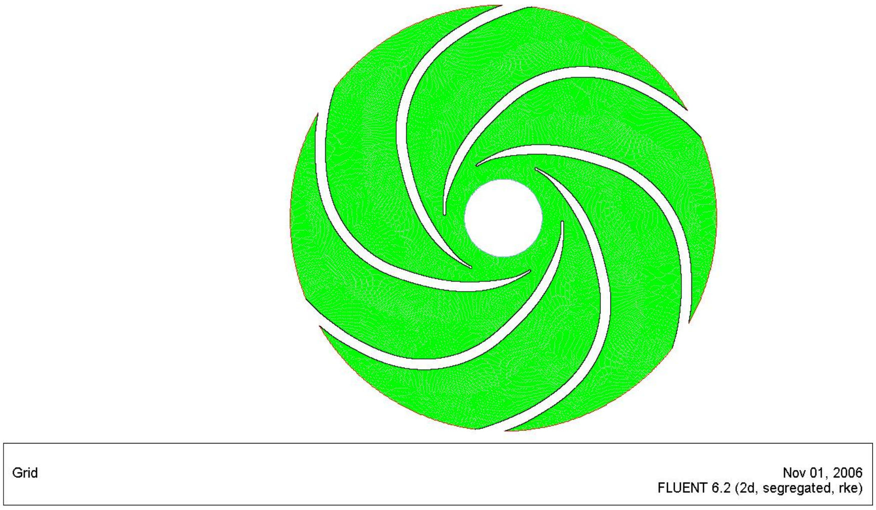


Figure 6.1: Final Grid after Importing into Fluent.

6.2.3 Materials

Next the fluid type was defined so that Fluent had the correct data for the coming calculations. All solution runs for this dissertation were completed with liquid water selected from the Fluent material library. By using the data contained in the library, there was no chance of having different values for the properties of water entered by mistake.

6.2.4 Solution Type

The most important and potentially most troublesome configuration option was next, and that was the selection of the solver to be used. After reading the recommendations given in the Fluent documentation, the Realisable k-ϵ viscous model was used with Enhanced Wall Treatments to correctly model the fluid interaction at the boundary layer of the blades. To ensure that this boundary layer is correctly resolved, the non-dimensional value y^+ needs to be less than 60 (Fluent Inc. n.d., 11.9.3). y^+ is the local Reynolds number of the flow at the wall for a given distance away. The closer the y^+ value gets to 1, the more accurate the modelling of the wall effects, because as y^+ heads towards 1, the local flow velocity at the wall drops towards zero.

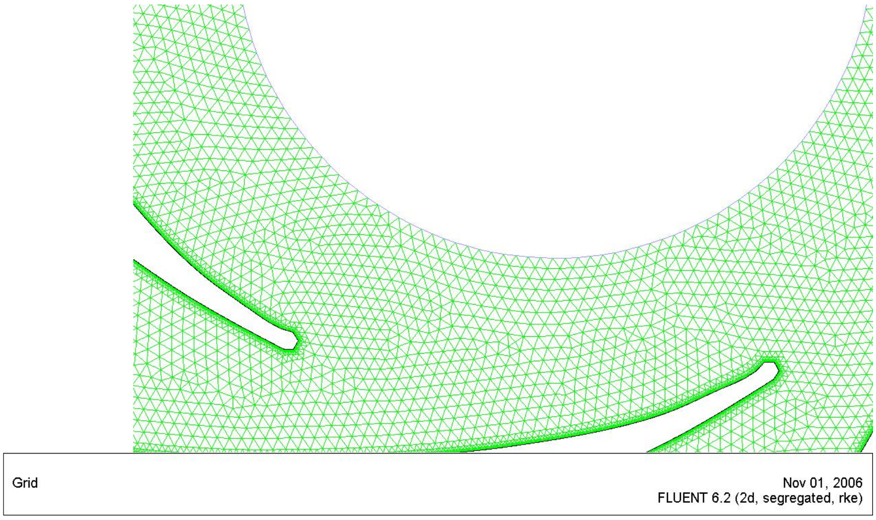


Figure 6.2: Refined Boundary Layer for low y^+ values.

All of the other solver options were left as standard, which results in a segregated two dimensional solution space with absolute velocity formulation and steady flow conditions.

The above solver settings are what the final run was configured for. The initial solver runs were completed with the standard k-ϵ viscous model with no wall effects, as recommended by Fluent. Once the solution had converged, the more advanced and slower to process solver options were enabled to refine the solution.

6.2.5 Boundary Conditions

Due to this model being an impeller, the boundary conditions needed to be setup to accommodate the rotation of the impeller to correctly simulate the flow conditions present. Without the rotation being allowed for, the flows present would not be indicative of the impeller in service.

The inlet condition for the solution was set as a pressure-inlet. This was set to an absolute pressure of 101,300 kPa, with a Turbulence Intensity of 10% and a Hydraulic Diameter of .02 metres. This Hydraulic diameter was selected as the size of the fluid inlet into each individual fluid passage.

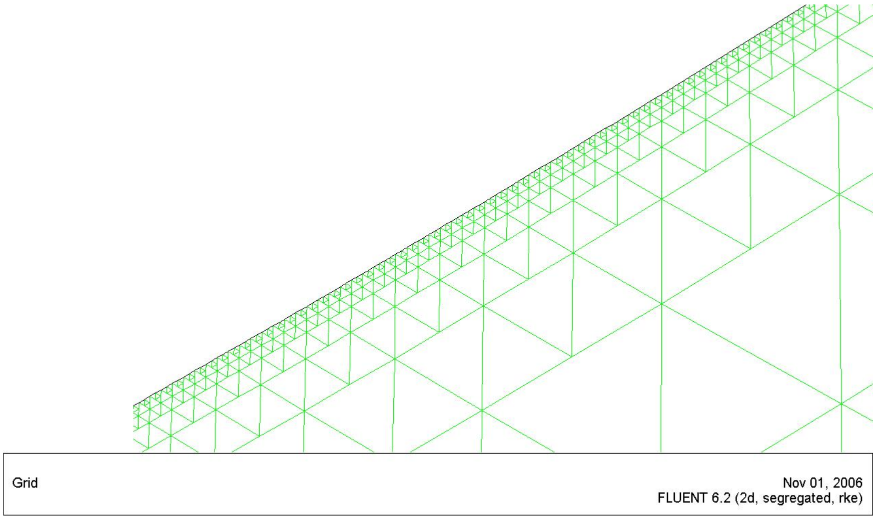


Figure 6.3: Closeup of wall showing boundary elements.

The outlet condition for the solution was set as a pressure-outlet. This was set to an absolute pressure of 101,150 kPa, with a Turbulence Intensity of 70%, with a Hydraulic Diameter of .02 metres. This hydraulic diameter was set the same as the inlet, due to the fact that on the real impeller, the cross sectional area is the same at the inlet and the outlet.

The impeller blades were set as moving walls, in a rotational motion, with a speed of 0 $\frac{rad}{s}$ relative to the adjacent cells. This was configured as per the recommendations from Fluent for the solution of rotation flows. The reasoning behind this is that for a stable solution, it is best to slowly increase the speed of rotation in approximately 10% increments up to full speed. By having the blade speed coupled to the fluid speed, it is only necessary to update the speed in one location, reducing the possibility of errors occurring.

6.2.6 Solution Monitors + Optional Features

With nearly all of the options that are needed configured, all that is left is to setup the monitoring of the residuals to provide an indication of how the solution is progressing. For these solutions, a basic monitor of the residuals is enough to give an idea of what is happening during the solution.

This solution has none of the optional features enabled, such as animations or some of the other monitors like force and volume as these monitors were not applicable to this solution. These features will become very important when running the full pump assembly as the forces monitor will enable visulisation of the system loading as the solution progresses.

As can be seen in Figure 6.4, the residuals plot shows the changes to the parameters by large spikes in the graph, but these settle down quickly as the solution converges rapidly at this point. The largest spikes were caused by the adaptation of the y^+ values via the automatic adaptation system in Fluent.

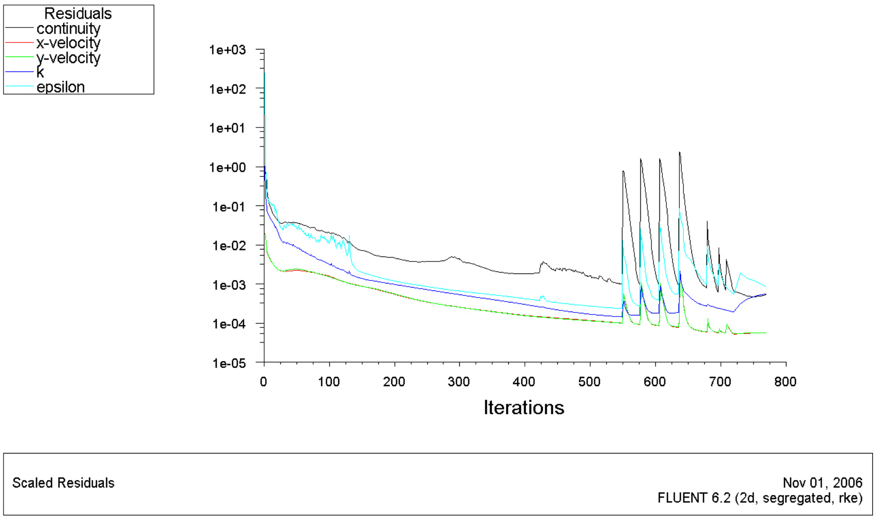


Figure 6.4: Graph of Residuals vs. Iterations from Fluent after the final solution was reached.

6.2.7 Iterations

The number of iterations necessary varies from solution to solution, but normally a well setup simple problem will solve in under 1000 steps. The complete solution for this dissertation took 770 steps, but this could be reduced by providing initial values to the solver that are closer to the values eventually calculated.

6.3 Results

The final results of the simulation run in Fluent confirmed some of the initial suspicions about the flow conditions that were likely to be present in the impeller, but there were also some flow effects that weren't expected. While this data doesn't represent what is happening in the actual pump impeller, it does validate some of the basic flow effects that will impact on the performance of the impeller. This first stage of the analysis provides a beginning for further research into the flow in these impellers, with the next stage being a full 3D model of the impeller to investigate 3D flow effects, rather than the 2D effects shown here.

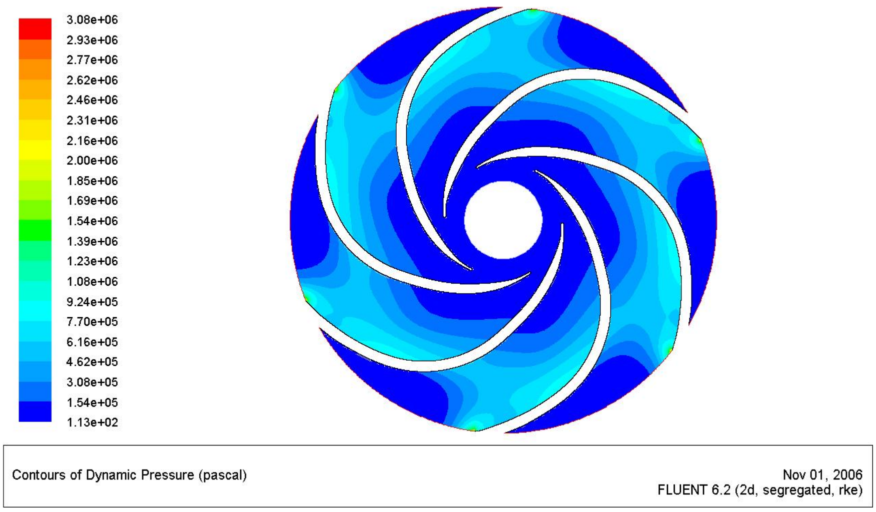


Figure 6.5: Contours of Dynamic Pressure

The dynamic pressure contour plot (Figure 6.5)of the impeller shows the pressures present in the impeller due to the static pressure in the system plus the pressure change due to the rotation of the impeller. In a perfect impeller, the rate at which the blade passage expands (which can be considered a diffuser increasing pressure and slowing the fluid) should be offset by the radial velocity the fluid gathers as it is accelerated by the spinning impeller. This complimentary effect should have the net result of producing no pressure rise across the impeller fluid passage.

In this case, it can be seen that from the inlet or eye of the impeller to approximately $\frac{1}{4}$ of the way down the suction side of the blade, and to approximately $\frac{1}{3}$ on the pressure side, the pressure does remain constant. After this point, the blade surfaces are more rapidly diverging, producing expansion that the fluid cannot follow. The region at the outlet where the pressure is shown to be the same as the inlet is a point where the flow has stopped due to re-circulation.

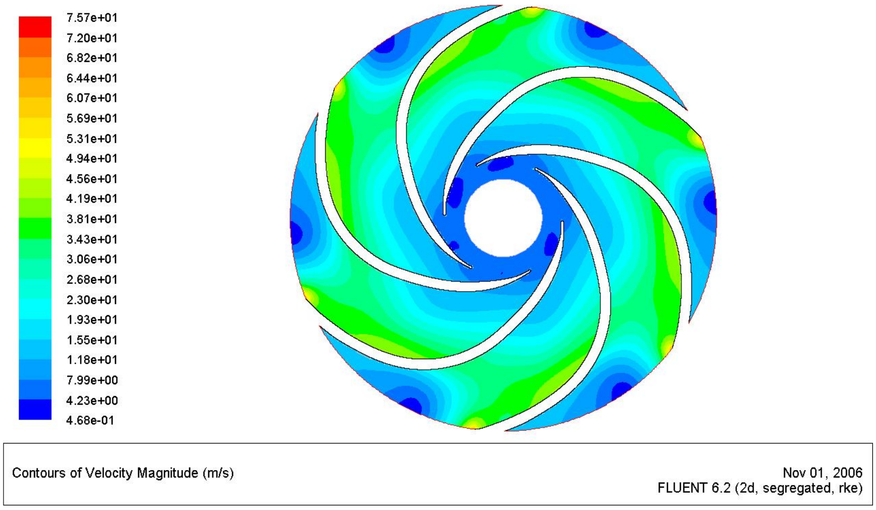


Figure 6.6: Contours of Velocity Magnitude

The velocity magnitude contour plot (Figure 6.6) shows the speed of the fluid in the impeller with regards to a fixed point. This plot shows the fluid accelerating from a low speed in the eye of the impeller to quite high speeds at the passage outlet. Again the outlet shows a point of re-circulation where the local flow has stopped. This plot also shows some separation at the eye of the impeller on the suction side of the blades. One aspect of the velocity magnitude plot versus the dynamic pressure contour plot that appears incorrect is that according to these plots, the pressure in the impeller rises as the velocity increases. The only explanation I can give for this is that the flow conditions just at the very end of the suction side of the blade is causing a very high pressure rise to occur as the fluid exits the computation domain, and the fluid is being affected by this (this pressure rise is shown in Figure 6.7). This hypothesis will be explored in the next stage of this modelling through my work.

The turbulence intensity contour plot (Figure 6.8)displays the turbulence present in the impeller as a percentage. The initial conditions used in the solution of this impeller agree well with the numerical solution, except at the outside end of the suction side of the blade. This very high local turbulence suggests that ending the computational domain right at the impeller exit can cause un-expected problems with the solution.

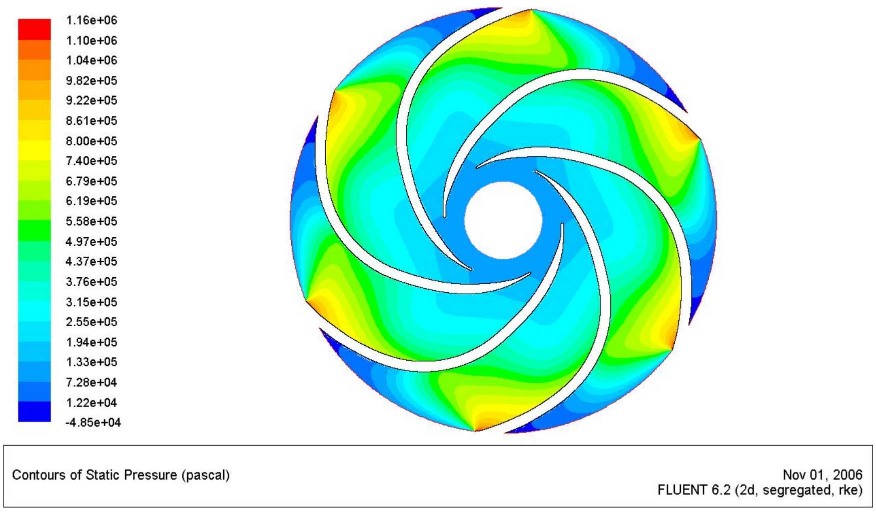


Figure 6.7: Static Pressure

A point to note is that in the eye of the impeller, it can be seen that the turbulence between the blades effects the fluid nearly all the way out to the inlet.

The wall y^+ contour plot (Figure 6.9) shows the values of y^+ close to the wall. The y^+ value increases quickly as the distance from the wall increases, and it can be seen that at the wall itself, the value is below 60 as recommended by the Fluent Documentation

Velocity vectors shown in Figure 6.10 and Figure 6.11 show the re-circulation present at the impeller outlet, along with the flow turning from the suction side of the blade about half way down as it separates. These plots give a good indication of the fluid behaviour and motion, and show that the blade design could use some improvement if the impeller was a 2D type like a stirrer in a tank.

The path line plot as shown in Figure 6.12 traces particles from the inlet to the outlet. It can be seen that these particles attempt to follow the curvature of the suction side of the impeller blade, but soon become detached. The path lines make a recovery near the end of the fluid passage and re-attach to the suction side, but I believe this is an error in the calculation due to the effect of the high pressure point just on the blade tip.

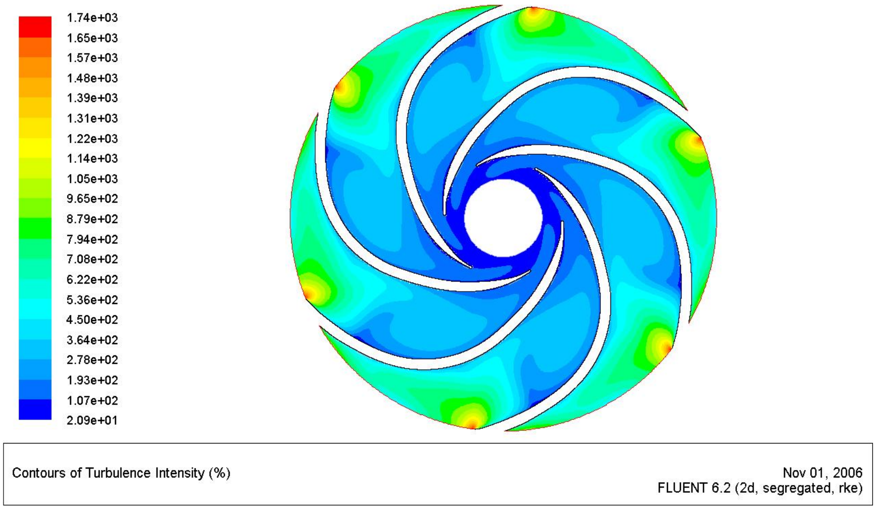


Figure 6.8: Turbulence Intensity

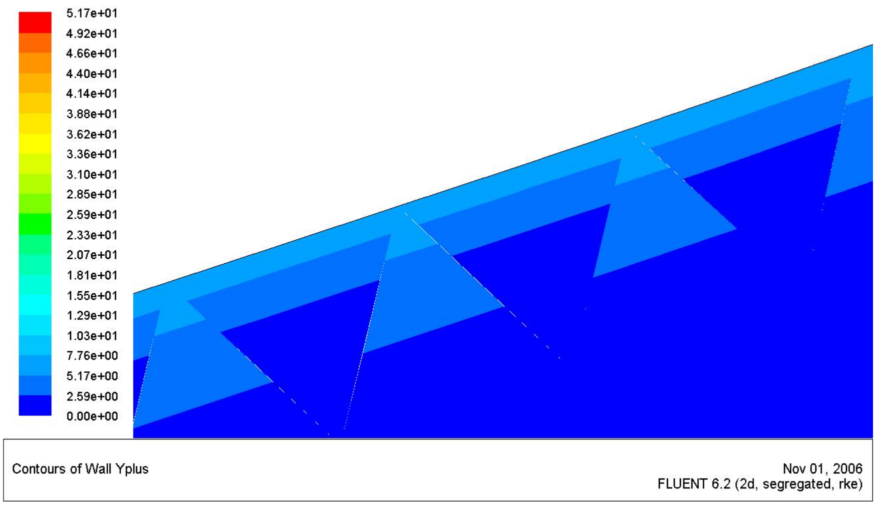


Figure 6.9: Contour of Wall y^+

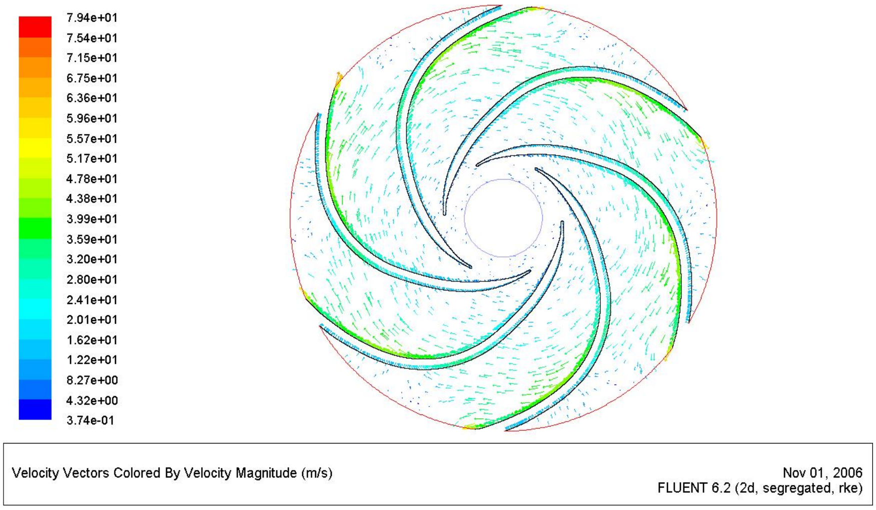


Figure 6.10: Velocity Vectors Coloured by Velocity Magnitude

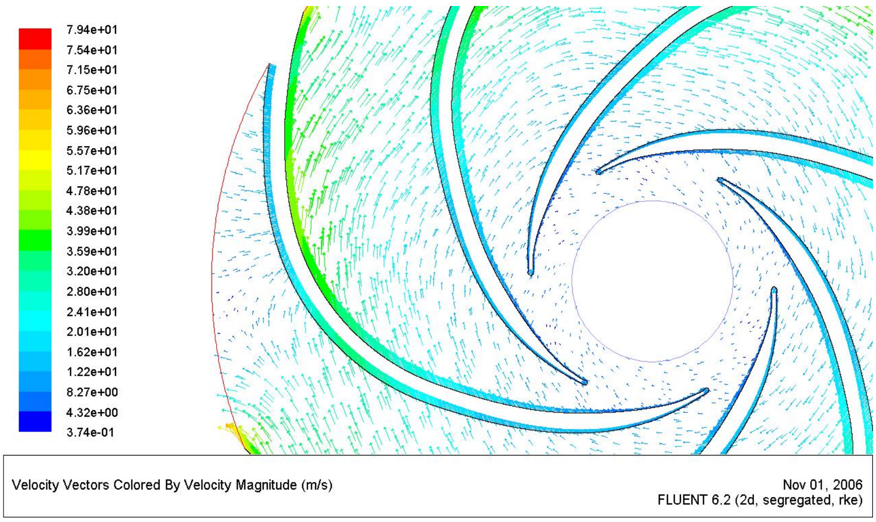


Figure 6.11: Velocity Vectors Coloured by Velocity Magnitude - Detail

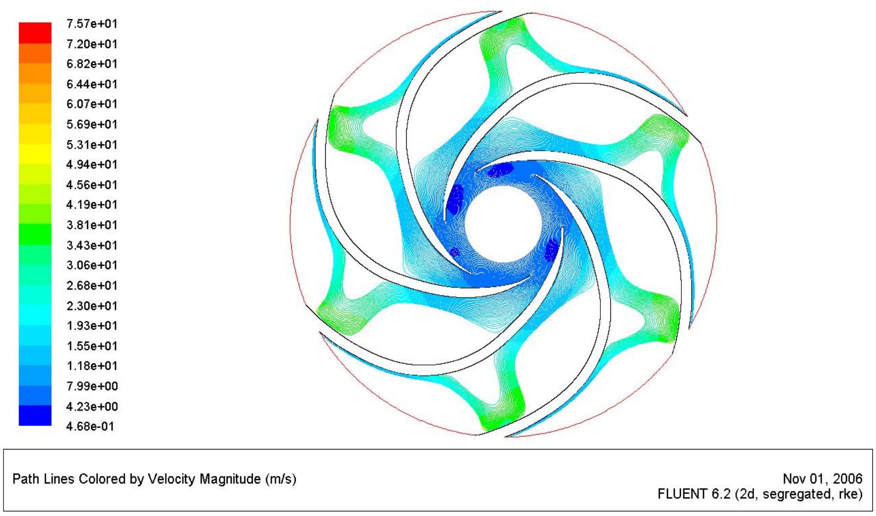


Figure 6.12: Path Lines Coloured by Velocity Magnitude

6.4 Chapter Summary

Overall, the solution gave some interesting points investigate. The effect of separation on the flows through the impeller was greater than was expected, and the re-circulation in a different location than first suspected. These points on the impeller will require special consideration when building the 3D geometry and mesh for later solutions to ensure there is enough resolution in the mesh to resolve changes in these regions.

What was unexpected was the abnormal pressures seen at the tip of the suction side blades. The reason for this will need to be discovered and the problem eliminated before the more complex simulations can be attempted.

Chapter 7

Conclusions and Further Work

When this topic was first suggested by my employer, I was doubtful meaningful results could be achieved with a 2D simulation. Now having completed the analysis of the impeller, I have found that even though the solution doesn't exactly match the flow conditions found in an actual impeller, the knowledge and techniques acquired during the research and execution of this dissertation will enable more complete and complex simulations to be built up. While some of the information pertaining to the designs and processes used by Tyco Pumping Systems/Flow Control could not be included due to confidentiality reasons, the generalised issues and procedures provide an insight into the production and design issues that need to be accounted when designing pumps.

7.1 Achievement of Dissertation Objectives

The following objectives have been addressed by this dissertation:

Background Research and Literature Review Chapter 2 presented background information about the current state of impeller production and design, as well as highlighting some of the solutions used to combat problems as they occur. These issues need to be accounted for and addressed in the design phase if possible when new product is developed. Chapter 3 presented some pertinent literature from the field, and although none of the literature had conducted an analysis in the same manner as this dissertation, the methodology, results and recommendations were of use when considering the solution path for this work.

Mathematical Modelling and Boundary Conditions Chapter 4 explained the maths behind the analysis, and also defined some of the pump and impeller characteristics.

Modelling And Meshing Chapter 5 covered the process of converting old drawings into a mesh suitable for solution in Fluent. Many difficulties were encountered during the modelling and meshing stage of the dissertation, and the solution heavily relies upon the work done at this stage. As each stage progressed, time was also taken to determine what would best suit TPS for their future CFD and design work, and it was determined that manual point measurement from the production castings is too time consuming, and that other methods such as laser scanning needs to be investigated.

Numerical Analysis Procedure and Results Chapter 6 brought together over a year of research and learning about pumping, CFD and fluid dynamics into the solutions presented here. While the analysis produced many results that were expected, many of the assumptions made were incorrect and these assumptions in some cases were long standing and thought to be correct. The insight into the flow conditions will enable better decisions to be made and will affect the path of development undertake at TPS.

7.2 Further Work

The scope for further work within this field is limitless. By progressing through to 3D analysis of the same impeller, the techniques and knowledge required for 3D analysis will be acquired. This knowledge can then be taken to the next level by applying the 3D impeller model to a 3D volute, and then learning the techniques and knowledge required for analysing the interaction between a stationary object and a rotating object with the same fluid medium.

It is that this stage that the work can continue into two distinct paths. The first path is to continue learning by applying the already acquired knowledge to multi-stage mixed flow, centrifugal and axial pumps, which the solving of each produce their own techniques and skills. The second path is to start on a validation process in which existing pump designs are modelled and simulated using CFD and then the results are compared to experimental test data. This process will enable CFD to be trusted as a verified method of testing the performance and capabilities of a new pump design.

At this stage, the processes come together again, with the small exception that the first path from before will need to undergo the validation process the same as the second path. Once the CFD method of design is validated for all of the pump types under development, it can be integrated with an automated selection and design package that will take inputs of the customer requirements and produce geometry data and CFD results to verify the combination is possible. This will then enable new designs to be evaluated before investing in tooling and spending time testing iterations.

This further work encompasses some of the goals that my employer set for this dissertation, and they believe that there is great potential in virtual development of pump designs.

In the future, CFD will become a resource for many companies, and Tyco Flow Control/Pumping Systems aims to be at the forefront of the market.

References

Addison, H. (1966), *Centrifugal and Other Rotodynamic Pumps*, third edn, Chapman and Hall, London, United Kingdom.

Bathe, K.-J., ed. (2005), *Proceedings of the Third MIT Conference on Computational Fluid and Solid Dynamics*, Elsevier, Cambridge, Massachussetts.

Brennen, C. E. (1994), *Hydrodynamics of Pumps*, third edn, Concepts NREC & Oxford University Press, London, United Kingdom.

Chu, C. (1978), 'Numerical methods in fluid dynamics', *Advances in Applied Mechanics* **18**, 285–331.

Ferziger, J. (1999), *Computational Methods for Fluid Dynamics*, Springer-Verlag, New York.

Kovats, A. (1964), *Design and performance of centrifugal and axial flow pumps and compressors*, Pergamon Press, Oxford.

Blanco-Marigorta et al. (2000), 'Numerical flow simulation in a centrifugal pump with impeller-volute interaction', *Journal of Fluids Engineering* .

Byskov et al. (2003), 'Flow in a centrifugal pump impeller at design and off-design conditions-part ii: Large eddy simulations', *ASME (American Soc. Mechanical Engineering) Journal of Fluids Engineering* **125**, 73–83. Viewed 3 May 2006.

Coutier-Delgosha et al. (2003), 'Experimental and numerical studies in a centrifugal pump with two dimensional curved blades in cavitating condition', *ASME (American Soc. Mechanical Engineering) Journal of Fluids Engineering* **25**, 970–978. Viewed 3 May 2006.

Fluent Inc. (n.d.), *Fluent 6.2 Documentation*, Fluent Inc.

Hamkins, C. & Bross, S. (2002), 'Use of surface flow visualization methods in centrifugal pump design', *ASME (American Soc. Mechanical Engineering) Journal of Fluids Engineering* **124**, 314–318. Viewed 2 May 2006.

Muggli et al. (2002), 'Cfd calculation of a mixed flow pump characteristic from shutoff to maximum flow', *ASME (American Soc. Mechanical Engineering) Journal of Fluids Engineering', volume =124, pages =* .

Weidong et al. (2003), 'Investigation of flow through centrifugal pump impellers using computational fluid dynamics', *International Journal of Rotating Machinery* **9**, 49–61. Viewed 8 May 2006.

Mestreau, E., Zhu, M., Kisielewicz, L., Löhner, R. & Goto, A. (1995), Finite element simulation of unsteady flow in complete turbomachines with rotating impellers using the ALE remeshing technique, *in* 'PUCA'95 Proceedings', Nihon ESI K.K., Tokyo, Japan.

Miner, S. (2000), 'Evaluation of blade passage analysis using coarse grids', *ASME (American Soc. Mechanical Engineering) Journal of Fluids Engineering* **122**, 345–348. Viewed 3 May 2006.

Stepanoff, A. J. (1957), *Centrifugal and axial flow pumps*, second edn, John Wiley & Sons, Inc., New York.

Stephen Lazarkiewicz & Adam T. Troskolanski (1965), *Pompy wirowe. (Impeller pumps.)*, Pergamon Press, Oxford.

Tezduyar, T., Behr, M. & Hughes, T. (1996), Finite element methods, *in* J. Schetz & A. Fuhs, eds, 'Handbook of Fluid Dynamics and Fluid Machinery', John Wiley & Sons, pp. 1272–1282.

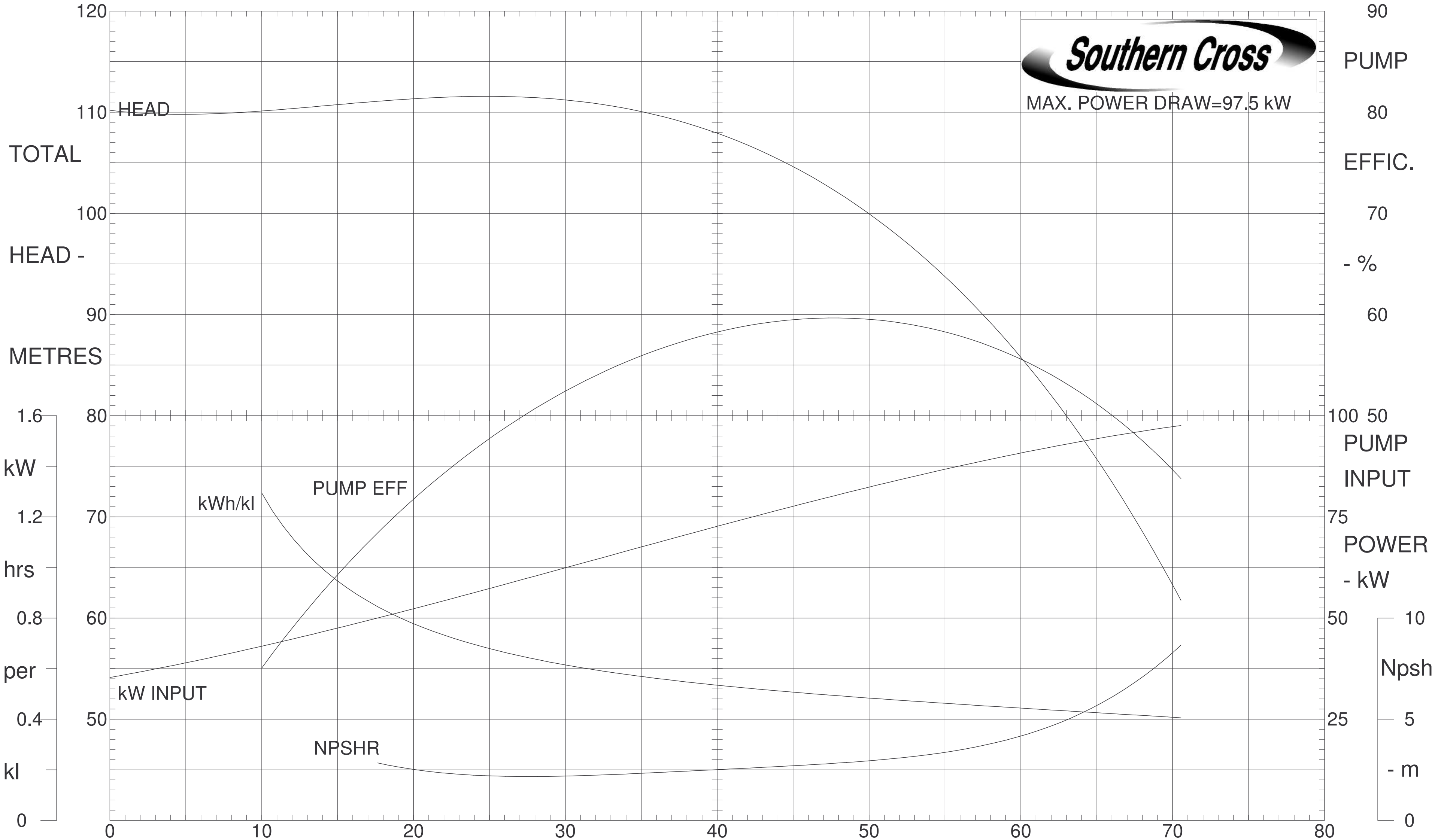
Turton, R. K. (1994), *Rotodynamic Pump Design*, Cambridge University Press, England.

Appendix A

Project Specification

Appendix B

Pump Curve



PUMP CONSTRUCTION - CAST IRON / CAST IRON
DATA REFER TO CLEAN COLD WATER

NO MOTOR EFFICIENCY DERATING APPLIED
PUMP PERFORMANCE TO AS2417 GRADE 2

Tyco Flow Control Pumping Systems	SOUTHERN CROSS 125x100-500 Impeller Dia 547mm, 1485 rpm MONARCH 110kW 4P TEFC MOTOR	CURVE No. SCP-ZZZ-1 DATE 02-11-06 ISSUE AT/AT

Appendix C

Fluent Contours

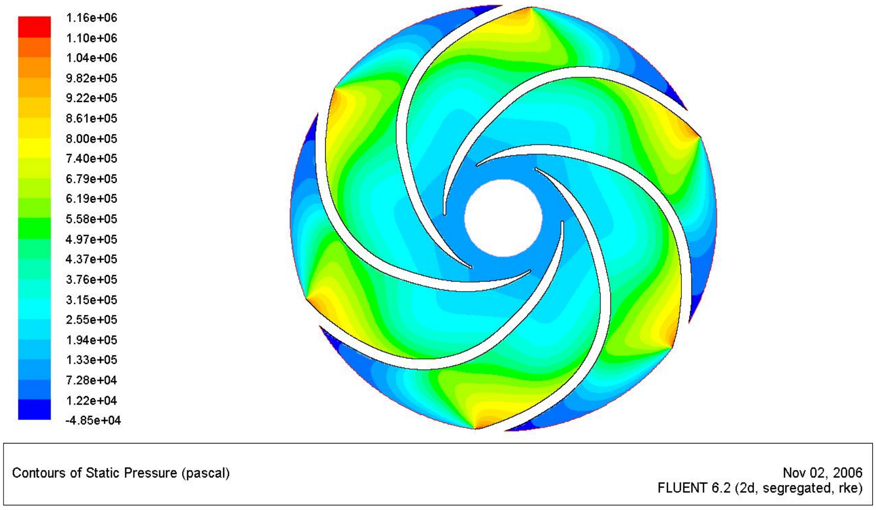


Figure C.1: Contours of Static Pressure

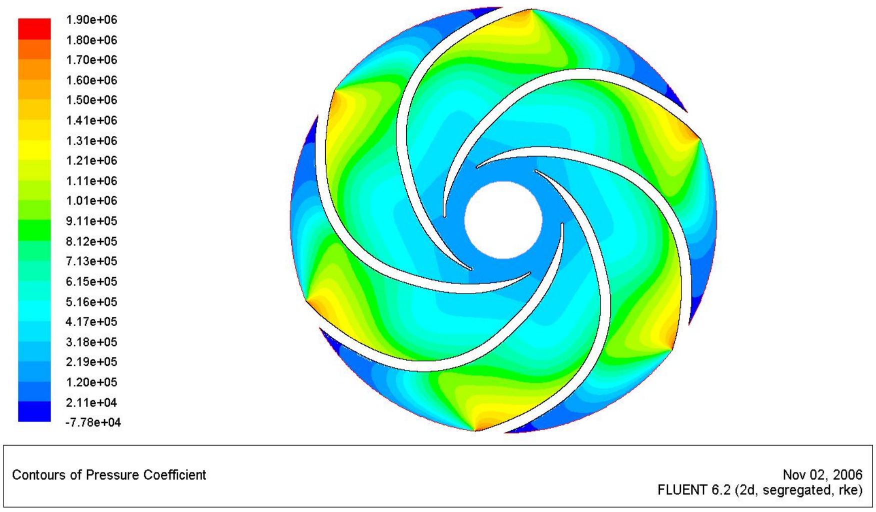


Figure C.2: Contours of Pressure Coefficient

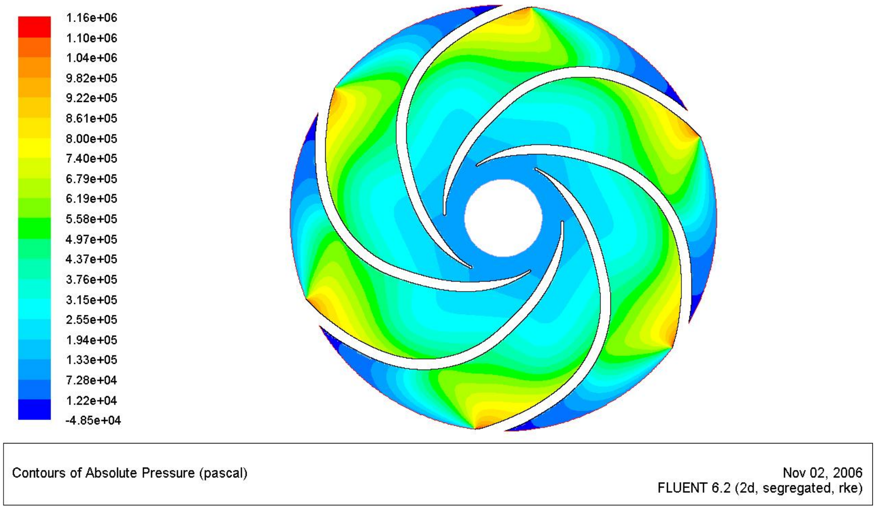


Figure C.3: Contours of Absolute Pressure

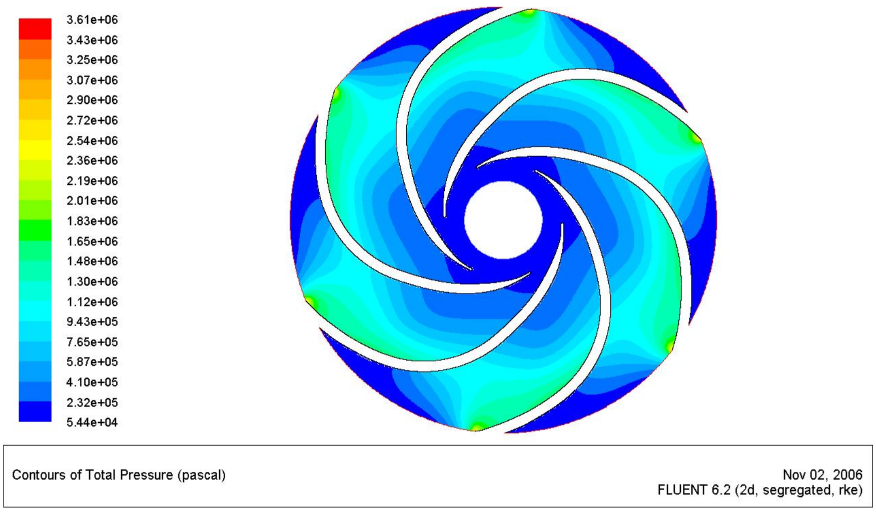


Figure C.4: Contours of Total Pressure

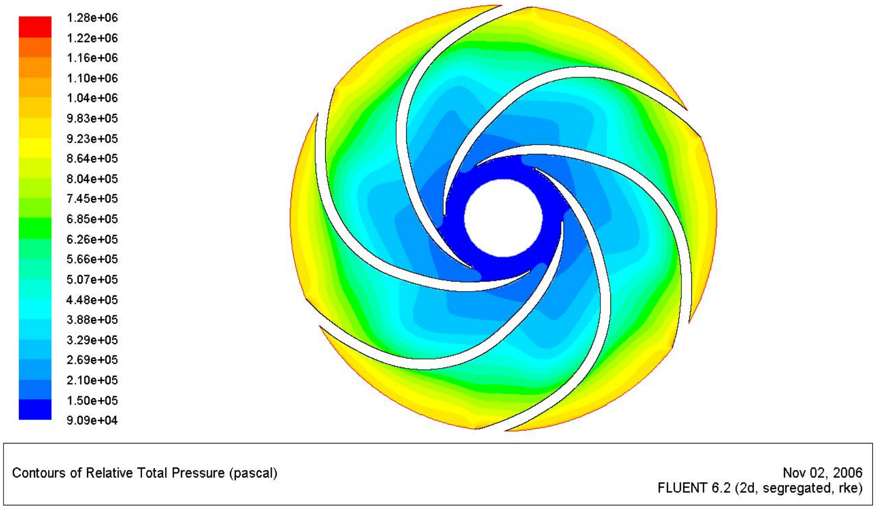


Figure C.5: Contours of Relative Total Pressure

Figure C.6: Contours of X Velocity

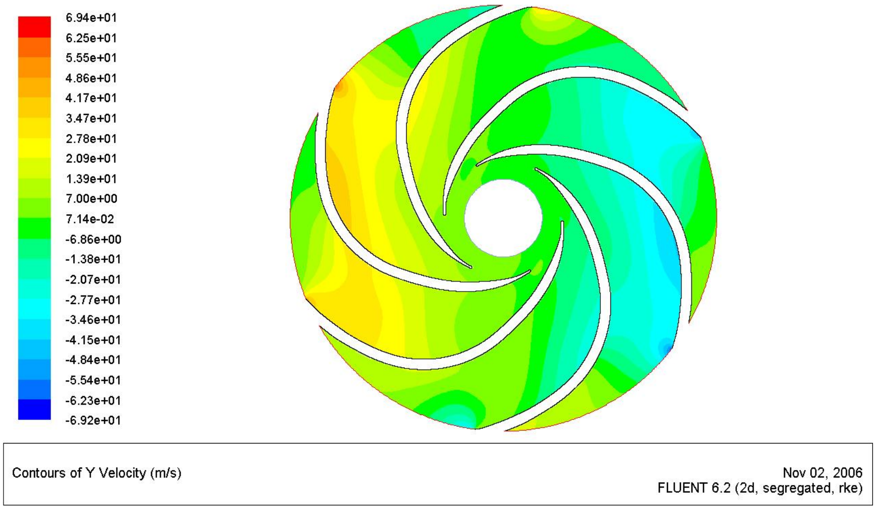


Figure C.7: Contours of Y Velocity

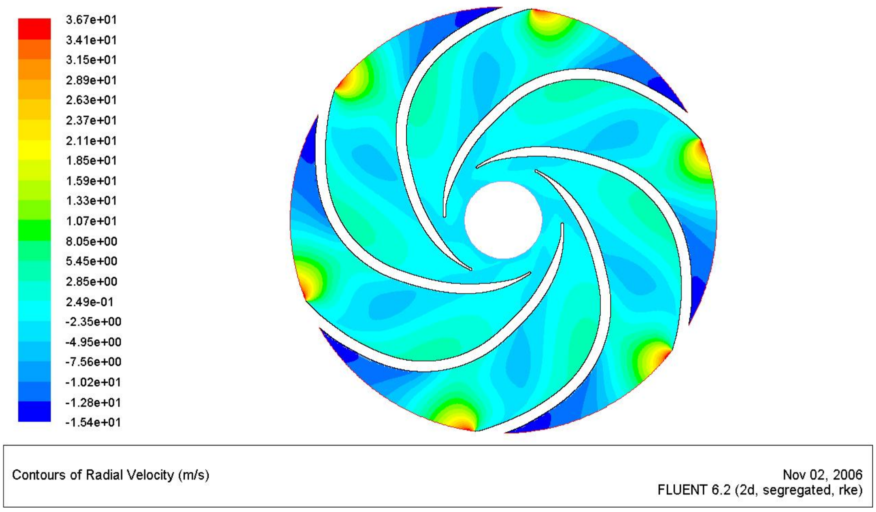


Figure C.8: Contours of Radial Velocity

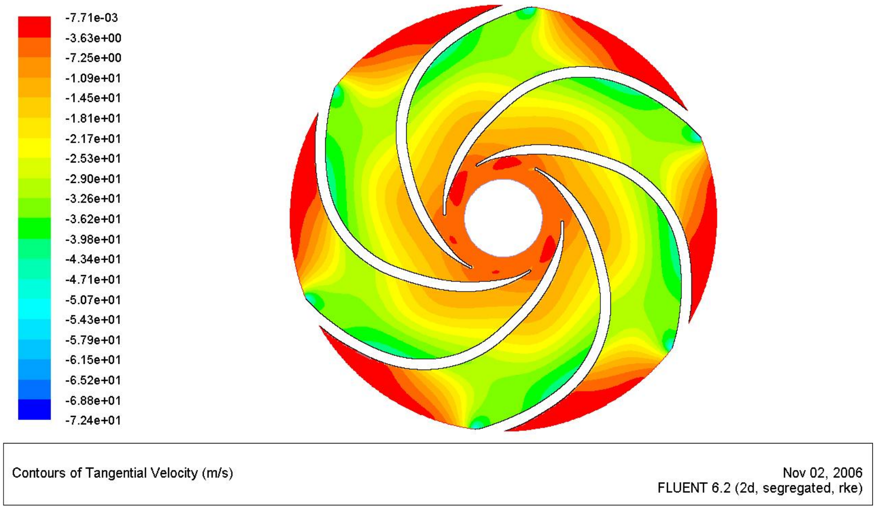


Figure C.9: Contours of Tangential Velocity

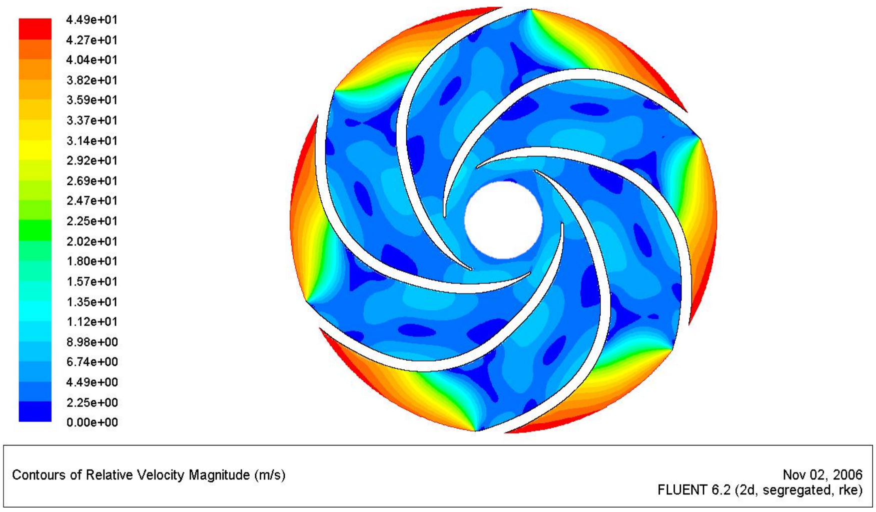


Figure C.10: Contours of Relative Velocity Magnitude

Appendix D

Company Brochure

COMPANY PROFILE

Southern Cross *is an Australian icon in the pumping and Irrigation industry having been established since 1871. The development of Southern Cross windmills and associated water storage and handling equipment played an important part in Australia's early development. The success of our early pioneering settlers basically came back to a dependence on one vitally important resource.... a reliable supply of good quality water. Today's modern industrial, community and agricultural developments have that same reliance, and even greater demand, for safe, clean and reliable supplies of water.*

Water handling equipment has been, and still is, the central development and manufacturing focus for Southern Cross throughout the 135 years, along with the marketing, distribution and servicing of the products not only throughout Australia but also to many developing countries worldwide.

In the early development the company also manufactured an extended range of products including such things as wool presses, steel railway sleepers, steam boilers, stationary and portable steam engines, dairy equipment, railway rolling stock and steam locomotives for both the Queensland and Commonwealth Governments. Quantities of munitions and appliances used during two world wars, and including hundreds of engines, pumps, air compressors, diesel marine engines and machine tools were manufactured for British Forces in North Africa, Americans for use in the East and for our own Army and Navy.

Currently the Southern Cross product range extends from small domestic water supply systems, through broadacre agricultural irrigation schemes, to specialised products for mining, heavy industry and community water supplies. Australia's leading range of ISO Standard centrifugal pumps, close coupled motorpumps, multi stage centrifugal pumps, submersible borehole pumps, computer controlled pump stations, and heavy duty water storage tanks and tankstands are all key products for the company. The irrigation product range includes small self-propelled turf and small crops irrigators, and high performance broadacre travellers.

Everflow Pumps *was established in Sydney in 1986 and emerged as a leading manufacturer and supplier of air operated diaphragm pumps, submersible pumps, turbine pumps and axial flow pumps for both the agricultural and industrial markets. Everflow Pumps was acquired in 2000 and later relocated to the new Southern Cross manufacturing facility in Witcott, Queensland. The broad range of Everflow turbine and submersible pumps is now manufactured and distributed from Withcott and marketed under the Everflow Pumps brand.*

Ph: 131 786 Fax: 1800 637 867
Web: www.southcross.com.au

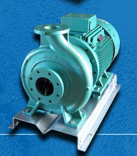
CENTRIFUGAL PUMPS

ISO SOVEREIGN CENTRIFUGAL PUMPS

- Irrigation
- Water transfer
- Building services
- End suction ISO2858
- Back pull-out-design
- Flow rates up to 250 litres / sec
- Pumping heads up to 160 metres

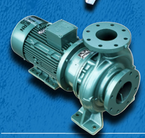
ISO-PRO CENTRIFUGAL PUMPS

- Mining
- Heavy industry
- Irrigation
- Belt, direct engine or electric motor drive
- Cast iron, ZF bronze or stainless steel
- Heavy duty shaft with oil lube bearings
- Flow to 250 litres / sec
- Heads to 160 metres



STAR-LINE CENTRIFUGAL PUMPS

- Irrigation
- Building services
- Pressure boosting
- Space saving design
- Cast iron construction
- Close coupled electric motor driven
- Flow to 100 litres / sec
- Heads to 160 metres



STAR-PRO CENTRIFUGAL PUMPS

- Heavy industry
- Commercial swimming pools
- Process water transfer
- Space saving design
- Cast iron, ZF bronze or stainless steel
- Close coupled electric motor driven
- Flow to 100 litres / sec
- Heads to 160 metres

MULTI-FLOW CENTRIFUGAL PUMPS

- Domestic
- Rural
- Water transfer & boosting
- Single and multi-stage design
- Flows up to 180 litres / min
- Heads up to 60 metres

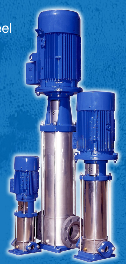
Ph: 131 786 Fax: 1800 637 867
Web: www.southcross.com.au

Southern Cross Everflow Pumps

MULTI-STAGE & HELICAL ROTOR PUMPS

VERTICAL MULTI-STAGE CENTRIFUGAL PUMPS

- Building services
- Irrigation
- Pressure boosting and transfer
- HVAC and boiler feed
- All wetted parts stainless steel
- Small pump footprint
- Flow rates to 21 litres / sec
- Heads to 250m



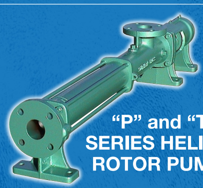
MULTI-STAGE AUTOMATIC PRESSURE SYSTEMS

- Large Residences
- Hotels & Motels
- Industry
- Constant Pressure
- Output to 18 litres per second
- Heads to 90m

"SS" HELICAL ROTOR PUMPS

- Water transfer over long distances
- Pressure boosting to high heads
- Stock watering
- Belt or direct drive
- Surface mounted or borehole installation
- Positive displacement type
- Flows to 2.2 litres / sec
- Heads to 120 m

"P" and "T" SERIES HELICAL ROTOR PUMPS



- Heavy duty design
- Industrial water transfer
- Chemical, effluent and process
- Surface mounted or borehole applications
- Positive displacement type
- Belt or direct drive
- Flows to 18 litres / sec
- Heads to 120 m

Ph: 131 786 Fax: 1800 637 867
Web: www.southcross.com.au

Southern Cross Everflow Pumps

TURBINE & SUBMERSIBLE PUMPS

SOUTHERN CROSS 4, 6 & 8 inch SUBMERSIBLE PUMPS

- Mine dewatering
- Industrial water supplies
- Municipal supply
- Irrigation
- Domestic water supplies
- All stainless or stainless steel / composite
- Capacities to 30 litres / sec
- Heads to 410m

SOUTHERN CROSS SUBMERSIBLE PUMPS 6 & 8 inch ZINC FREE BRONZE

- Industrial
- Municipal water supply
- Commercial irrigation
- All bronze construction
- Suit 150mm or larger bores
- Capacities to 55 litres / sec
- Heads to 300m

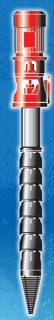
EVERFLOW SUBMERSIBLE PUMPS 6 to16 inch

- Mine dewatering
- Industrial water supplies
- Municipal supply
- Irrigation
- Domestic water supplies
- Heavy duty cast iron, zinc free bronze or stainless steel.
- Capacities to 320 litres / sec
- Heads to 230m

SOUTHERN CROSS LINE-SHAFT TURBINE PUMPS

- Irrigation
- Municipal water supply
- ZF bronze & stainless steel construction
- Shaft driven
- Borehole, well or riverbank
- Capacities to 50 litres / sec
- Heads to 200m

EVERFLOW LINE SHAFT TURBINE PUMPS



- Irrigation
- Community water supplies
- Borehole, well or riverbank
- Electric, vertical belt or right angle gear drive
- Oil or water lubricated shaft
- Outputs to 750 litres/sec
- Heads to 300m

EVERFLOW AXIAL FLOW PUMPS

- Irrigation
- Flood lifting
- Well, dam or riverbank
- Single or multi-stage
- High volume, low head
- 150 to 1000mm propeller
- Vertical/inclined installation
- Capacities to 5000 litres/sec
- Heads to 25m

Ph: 131 786 Fax: 1800 637 867
Web: www.southcross.com.au

Southern Cross Everflow Pumps

TRAVELLING IRRIGATORS

SX-10 IRRIGATOR
- Turf Watering
- Greens
- Small Sporting Fields
- 55m x 1 inch hose
- 245 - 345 kPa
- 0.5 - 1.5 litres/sec
- Wetted width 28 - 40m

SX-20 IRRIGATOR
- Parks and Gardens
- Small Crops
- Turf Irrigation
- Sporting Fields
- 67m x 1 1/2 inch hose
- 400 - 500 kPa
- 2.7 - 3.9 litres/sec
- Wetted width 40 -55 m

SX-30 IRRIGATOR
- Large Sporting Fields
- Dust Suppression
- Small Crops ● Fairways
- Parks & Gardens
- Market Gardens
- 100m x 2 inch hose
- 400 - 620 kPa
- 1.6 - 6.0 litres/sec
- Wetted width 47 - 65 m

SX-2500 IRRIGATOR
- Commercial Crops
- Dust Suppression
- Feedlots and Pasture
- 100, 150 or 200m x 2 1/2 or 3 inch hose
- 400 - 620 kPa.
- 5 - 25 litres/sec
- Wetted width 70 - 120 m
- Up to 3 ha per run

SX-3500 IRRIGATOR
- Broadacre
- Feedlots and Pasture
- Dust Suppression
- Fodder Crops
- 150m x 3 1/2 or 4 inch or 200m x 3 or 3 1/2 inch hose
- 400 - 620 kPa
- 11 - 25 litres/sec.
- Wetted width 80 -120 m
- Up to 3.4 ha per run

TX-550 IRRIGATOR
- Broadacre
- Plantations
- Stock Feed and Pasture
- 200m x 3 1/2, 4 or 4 1/2 inch or 300m x 4 or 4 1/2 inch hose
- 500 - 620 kPa
- 10 - 44 litres/sec
- Wet width 90 -144m
- Up to 5.8 ha per run

HYDRANTS & OUTLETS
- Cast Aluminium
- Lightweight, Sturdy ● 3, 4 and 5 inch
- Connect Irrigator Hoses and Sub-Mains to Underground Mains

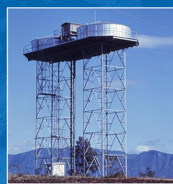
Ph: 131 786 Fax: 1800 637 867
Web: www.southcross.com.au

Southern Cross Everflow Pumps

TANKS, TANKSTANDS, WINDMILLS & PRESSURE SYSTEMS

PRE-FABRICATED LINER TANKS
- Commercial
- Stock watering
- Effluent storage
- Extensive range from 9 kilolitre
- Simple to erect
- Plastic liners
- Pre-fabricated galvanised steel

PRE-FABRICATED SQUATTERS TANKS
- Industrial ● Mining
- Air conditioning
- Fire services
- Municipal water storage
- Heavy duty bolted strip seal type pre-fabricated for easy transport and on-site assembly
- Extensive range from 9 to 3000 kilolitre



GALVANISED STEEL TANKSTANDS
- Municipal ● Rural
- Domestic
- Suit most steel and poly tanks
- Extensive range to suit varying tank sizes and elevation
- Heavy duty pre-fabricated
- All steel components hot dip galvanised

WINDMILLS & PUMPS
- Remote area water supply
- Environmentally friendly
- All weather operation
- Economical water supplies
- Durable and low maintenance
- 316 stainless steel windmill pumps and accessories

AUTOMATIC WATER PRESSURE SYSTEMS
- Domestic water supplies
- Light commercial water pressure boosting
- Domestic irrigation
- Pressure tank or "Presscontrol" operation
- Stainless steel or cast Iron construction
- Flows to 180 litres /min
- Heads to 90 m

Ph: 131 786 Fax: 1800 637 867
Web: www.southcross.com.au